ENGINSOFT

futurities

The Simulation Based Engineering & Sciences Magazine

Year 20
03
Autumn
2023

SPOTLIGHT
The art of sailing

- Editor's Note

The autumn issue of Futurities places its **Spotlight** on sailing and maritime and looks at how engineering is supporting the move to more sustainable transport in this sector.

Sustainability continues to be a key theme across all industry sectors, with its necessity having been sharply highlighted by the various extremes of weather experienced across the globe in the last three months – from raging wildfires, to devastating floods, from extreme heat waves to drought and freak hailstorms. If public pressure were not enough to encourage more attention to this issue, then Nature herself is beginning to loudly add her voice to the chorus.

As mentioned in the summer issue's *Editor's Note*, pressure is growing on engineers to innovate and find creative solutions to address the various aspects of the overall sustainability challenge – both in the short term as stop-gaps, and for the long term to replace fossil fuels and make industrial and commercial activities cleaner, more efficient and generally less harmful. Our **Spotlight** includes an in-depth **Face-to-Face** interview by the University of Tor Vergata with Ignazio Maria Viola and Ubaldo Cella of the *Journal of Sailing Technology* on the rising importance of simulation in the nautical sector, while there is also a detailed article on the use of CFD for optimizing hull design to achieve greater efficiency and lower ship resistance. We also included a few articles from the UE's *Horizon Magazine* on other developments in the sector including the development of modular batteries, green hydrogen and new sail and boat shapes.

This edition's **Technology Transfer** section takes a closer look at the challenges in variable-resolution particle methods of particle-based CFD (computational fluid dynamics), and discusses some of the existing methods compared to the new approach taken by moving particle simulation (MPS).

There are three articles in our **Know-how** section in this issue, one of which examines the optimization of the paint application process by robots inside a spray booth to reduce overspraying – a key problem in the process, while another explores the use of multi-camera DIC (digital image correlation) for measurement in testing departments in order to characterize materials or test mechanical structures.

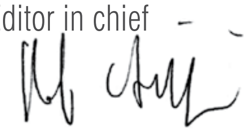
In the **Research & Innovation** section, we again have some success stories from the EU's FF4EuroHPC: a case study from CDR Pompe on the optimization of four different sizes of magnetic-drive chemical-process mechanical pumps, widely used in the chemical, pharmaceutical and nuclear industries where aggressive or hazardous fluids are handled; and a case study from Eunice Wind and FEAC Engineering on the challenges of creating an accurate simulation of a Eunice wind turbine's aerodynamic performance in various operating scenarios in order to improve its operational performance.

This section the magazine also examines the OSCAR project from the EU's EIT Manufacturing body, which was a collaborative undertaking over a year to define a technological solution for repairing metal components using directed energy deposition (DED) additive manufacturing (AM) in an effort to help industrial manufacturers define an automated or semi-automated solution to 3D scan a part with a partially unknown shape in order to automatically generate an AM repair program to return that part to the desired shape.

Another interesting article included in **Research and Innovation** this month concerns the EU's LIFESAVER project, coordinated by EnginSoft. This project aims to create a system that can replicate the placenta's behaviour during the first three months of pregnancy and use it to forecast the passage of various chemicals between mother and foetus through the placenta in order to detect concentrations of harmful substances such as antibiotics, antivirals, hormones or microplastics. The digital twin will enable this movement to be predicted and measured for the first since. This is because testing on pregnant women and their foetuses is obviously prohibited, and animal testing is not useful because their placentas behave differently.

I wish you pleasant reading!

Stefano Odorizzi
Editor in chief



"...pressure is growing on engineers to innovate and find creative solutions to address the various aspects of the overall sustainability challenge...

Futurities
Year 20 n°3 - Autumn 2023
To receive a free copy of the next magazine, please contact our Marketing office at: marketing@enginsoft.com

EnginSoft S.p.A.
24126 BERGAMO c/o Parco Scientifico Tecnologico
Kilometro Rosso - Edificio A1, Via Stezzano 87 • Tel. +39 035 368711
50127 FIRENZE Via Panciatichi, 40 • Tel. +39 055 4376113
35129 PADOVA Via Giambellino, 7 • Tel. +39 049 7705311
72023 MESAGNE (BRINDISI) Via A. Murri, 2 - Z.I. • Tel. +39 0831 730194
38123 TRENTO fraz. Mattarello - Via della Stazione, 27 • Tel. +39 0461 915391
10133 TORINO Corso Marconi, 20 • Tel. +39 011 6525211

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

Futurities is a quarterly magazine published by EnginSoft SpA

COMPANY INTERESTS
EnginSoft GmbH - Germany
EnginSoft UK - United Kingdom
EnginSoft Turkey - Turkey
EnginSoft USA
Particleworks Europe
www.enginsoft.com

CONSORZIO TCN www.consorziotcn.it • www.improve.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it
AnteMotion antemotion.com

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: marketing@enginsoft.com

RESPONSIBLE DIRECTOR
Stefano Odorizzi

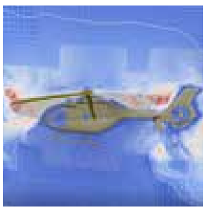
PRINTING
Grafiche Dalpiaz - Trento

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

Contents

RESEARCH & INNOVATION

PRODUCT PEEKS



SPOTLIGHT

SPOTLIGHT

The art of sailing

How engineering and simulation are contributing to cleaner shipping

The *European Maritime Transport Environmental Report*, published in 2021 by the European Environment Agency and the European Maritime Safety Agency in collaboration with the European Commission, found that:

- Maritime transport in the EU contributes to 13.5% of all greenhouse gas emissions from transportation in the region.
- Ships that arrived at European ports in 2019 emitted around 1.63 million tonnes of sulphur dioxide (SO_2), constituting around 16% of the total global SO_2 emissions from international shipping, contributing to air pollution.

Many of maritime transport's environmental effects could be resolved through the use of clean hydrogen, which is generated by splitting water through electrolysis powered by renewable energy sources. However experts agree that significant investment is still required to make this option commercially viable fast enough to meet the Paris climate goals.

The maritime sector is beginning to respond. The International Maritime Organization (IMO) plans to create and implement steps by 2025 to achieve its new GHG emission-reduction targets . In the interim, measures such as "slow steaming" where ships operate at slow speed thereby significantly reducing their fuel consumption, combined with route optimization, can help to reduce emissions per ship by up to 5% .

Engineering and simulation can play a significant role in reducing shipping's environmental impact, such as by:

- designing more energy-efficient ships
- optimizing hull shapes to minimize ship resistance and increasing fuel efficiency
- designing monitoring and management systems to model and improve energy consumption and efficiency
- studying navigation routes and using data on weather, currents, winds and ships to optimize routes
- innovating technologies for fuel and propulsion

In this edition, *Futurities* turns its **Spotlight** on the shipping sector and studies the use of CFD (computational fluid dynamics) to optimize hull design; explores the development of new sail and boat shapes for wind-assisted shipping; investigates the development of modular batteries for use across all types of vessels, and of green hydrogen fuel cells to power inland vessels; and we carry a face-to-face interview by the University of Tor Vergata with Ignazio Maria Viola and Ubaldo Cella of the *Journal of Sailing Technology* on the rising importance of simulation in the nautical sector.

References

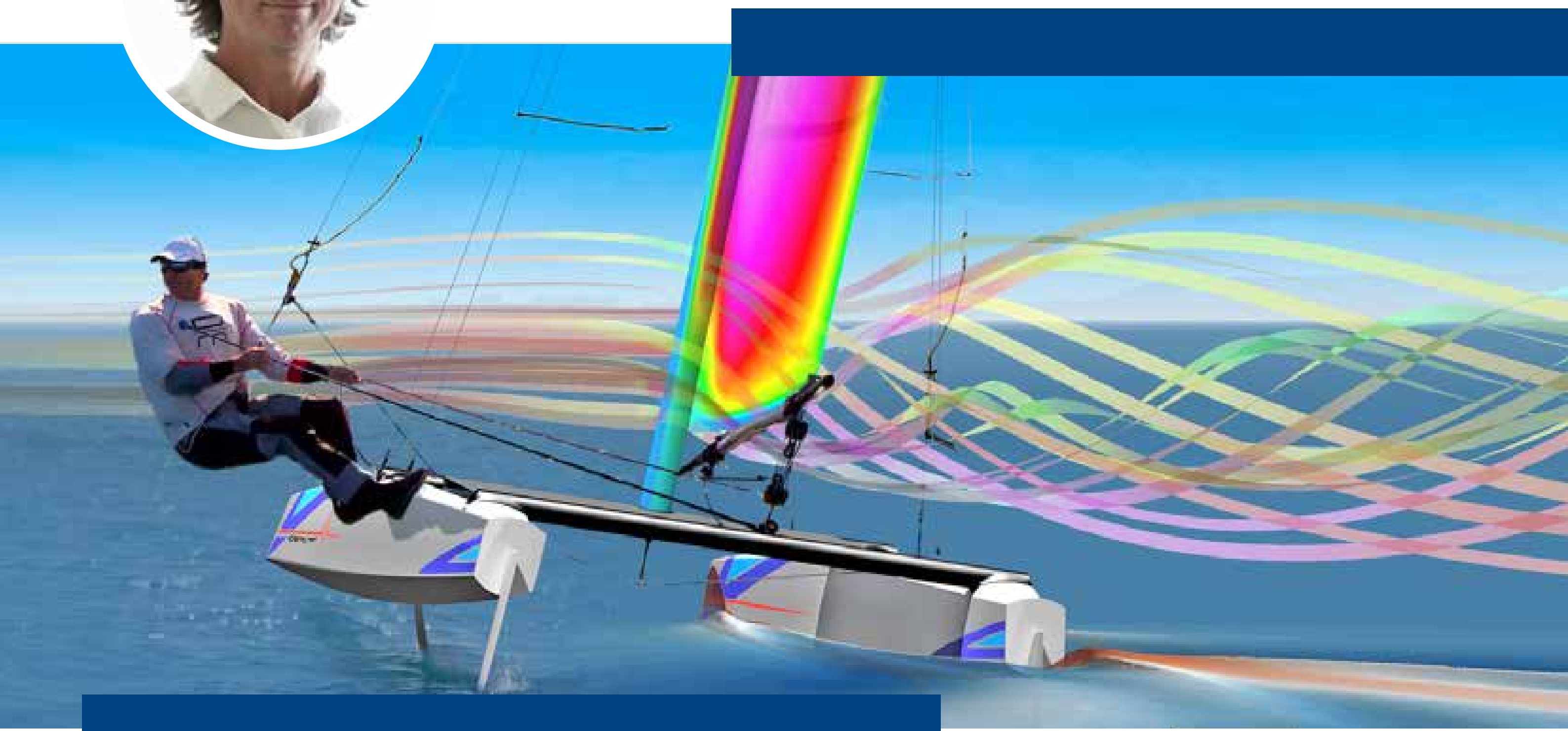
[1] www.eea.europa.eu/highlights/eu-maritime-transport-first-environmental

[2] climate.ec.europa.eu/eu-action/transport/reducing-emissions-shipping-sector_en

[3] theconversation.com/five-ways-the-shipping-industry-can-reduce-its-carbon-emissions-94883

face to face with Ignazio Maria Viola and Ubaldo Cella

Journal of Sailing Technology



The growing importance of numerical simulation in the nautical sector

By Marco Evangelos Biancolini
University of Rome Tor Vergata

The *Journal of Sailing Technology* (JST) is the definitive source for keeping abreast of the latest trends and technologies that are propelling the nautical sector. JST is a free, open access, and peer-reviewed journal that has become a global reference for the future of sailing thanks to its compelling content and unquestionable authority, which is why we decided to interview Ignazio Maria Viola, Editor-in-Chief, and Ubaldo Cella, member of the editorial board.

It is widely recognized that computer-aided engineering (CAE) has been playing an increasingly significant role in the marine industry in recent years. On the other hand, not many know that mesh morphing is an essential technology that can greatly improve computer simulation.

It offers numerous advantages in the geometric parameterization of design problems, drastically reducing development time and cost. RBF Morph is a pioneer and leader in RBF (radial basis functions) tools, and a long-standing Ansys partner.

Ignazio Maria Viola, Professor at the University of Edinburgh's School of Engineering's Institute for Energy Systems, and Ubaldo Cella, a researcher in the "Mario Lucertini" Department of Enterprise Engineering at the University of Rome Tor Vergata and a Senior Engineer at RBF Morph, offer an in-depth look at the rapid innovation that is disrupting this sector. Cella, an experienced recreational and competitive sailor, also won an award in the 2016 Ansys "Hall of Fame" competition for a study on a high-performance catamaran.

The "Mario Lucertini" Department of Engineering at the University of Rome Tor Vergata includes rbfLAB, a research laboratory focused on the advanced use of RBF in the most diverse fields and applications of engineering, from shape optimization for automotive or aerospace to the study of fluid-structure interaction in the biomedical field. DemoFly - a rbfLAB-led research project focused on the design of sport catamarans - aims to develop a wide-ranging design methodology that combines the search for the best overall boat configuration with the application of high-fidelity tools to optimize sails and appendages. DemoFly entails the development of overall performance analysis methodologies, the optimization of the various components, and the validation of the tools through the construction of prototypes and experimental tests [1]. The University of Edinburgh hosts VOILAb, a team of researchers focused on fluid mechanics, operating within the Institute for Energy Systems. This group is under the leadership of Prof. Ignazio Maria Viola (voilab.eng.ed.ac.uk).

How is JST positioned in the sailing industry?

Ignazio Maria Viola: JST strives to be a repository of the best papers and an archive of the advances in the science and technologies of sailing, including sailboats as well as any other application related to sails and wind propulsion. The journal also aims to support the research community in rigorously conducting and reporting their studies, contributing to improving the quality of research in the field.

What role does CAE play in the nautical sector today?

Ubaldo Cella: The sailing industry has evolved enormously in recent years, and the role of CAE has become increasingly significant. CAE incorporates leading design tools used for major sailing competitions, in particular the application of aeronautical skills for the America's Cup (AC). This process began several decades ago. One notable example was in the 1992 AC, when Raffaele Marazzi, an aeronautical engineer coming from the Italian aircraft manufacturer Aermacchi, used CFD (computational fluid dynamics) to design the keel and winglets of the Moro di Venezia. Today, the use of numerical analysis technologies is standard practice and confidence in their accuracy has drastically reduced the need for experiments during the design process. CAE methods are now commonly used in both cruising and racing boat design, making them an indispensable tool for the industry.

We are seeing a growing interest in open source software for design. What is the status of commercial versus open source tools and which ones are most used in the nautical sector?

Ignazio Maria Viola: The most demanding computational simulations are those related to fluid dynamics. They can calculate the aerodynamic forces exerted by air on the sails, and by water on the hull and appendages. Commercial software is often preferred for more complex simulations, for example, to calculate the hydrodynamic forces on a vessel free to heel, trim and sink, due to its robustness and ease of use. In contrast, an open source code such as OpenFOAM is often used when the modelled physics is simpler, such as for the air around a sail that is considered rigid. Open source codes are also preferred to commercial software when the accuracy of the simulation is critical and high spatial and temporal resolutions are required, involving several thousands of parallel processors. This is often the case in fundamental research, where OpenFOAM is increasingly being used in the scientific community.

How is RBF Morph positioned in this field and what is its role in the yacht design process?

Ubaldo Cella: Mesh morphing techniques offer several advantages for geometric parameterization of design problems, including: the ability to work directly on numerical domains; consistency of numerical analysis solutions (eliminating the need for remeshing of discretized domains); and robustness of procedures. Software based on radial basis functions, like the RBF Morph solver, offers additional advantages, such as highly efficient parallelization and high-quality morphing actions. RBF Morph's user interface offers great flexibility in configuring complex parameterizations, making it a key component in developing design methodologies, especially where numerical optimizations and automated analysis workflows are involved. These advantages have been amply demonstrated in a wide range of scientific literature addressing design problems in various engineering fields. Therefore, RBF Morph's efficiency, robustness, and speed of configuration make it a highly effective tool for engineers striving to optimize design parameters and achieve high-quality results [2, 4].

Could you describe the integration of reduced order models (ROM) and digital twins in the design of a sailboat and during racing?

Ignazio Maria Viola: Reduced order models such as potential flow theory implemented in panel codes are very common in both sail aerodynamics and hull hydrodynamics. They have been widely used in yacht design since the 1970s and have proven successful in high performance sailing. Digital twins, used to perform time progressing numerical simulations of boat dynamics, are commonly used in America's Cup design and are crucial for designing boats that manoeuvre quickly, for example.

What observations do you have regarding multi-objective and multi-physics shape optimization?

Ubaldo Cella: A numerical optimization approach to design is key to exploring the boundaries of a design problem. When combined with multi-disciplinary analysis tools, optimization procedures can significantly expand the possibilities of exploration. However, a common mistake is to view such methods simply as tools that automatically provide the best solution at the touch of a button. When used with competence and awareness, numerical optimizations are

powerful tools that can produce outstanding results. However, their application requires appropriate skill and experience; otherwise, they become an expensive way to generate a solution that is not necessarily the best. Developing numerical optimization environments also requires specialized knowledge in several fields, such as parametric solid modelling, CFD, FEM (finite element method), programming, and mesh morphing. The theory of numerical optimization is a vast discipline, and a thorough understanding of the advantages and limitations of the numerous algorithms available is crucial. In combination these factors make it particularly difficult to develop efficient optimization strategies and the results are always highly user-dependent. Therefore, engineers must have diverse skills and a deep understanding of the design problem they are addressing [3].

What are the main trends regarding the design methodologies we talked about earlier?

Ignazio Maria Viola: The rapid increase in available computing resources makes it possible to perform more complex simulations with greater ease and accuracy.

A good example of simulations that are becoming more accessible and affordable in terms of computation are fluid-structure interaction simulations. These simulations necessitate mesh modification or the remaking of a new mesh at each iteration to reflect the changed solid body geometry. We expect to see a substantial increase in the number of journal articles on these types of simulations for both sail aerodynamics and foil hydrodynamics in the coming years. On the other hand, there are already a number of papers showing advances in machine learning for yacht design, which is likely to become another major trend in the coming years. Finally, across the ship building industry, there is an increased use of model reduction methods in which high-fidelity simulations are used to inform lower fidelity models that provide the solution for a larger set of parameters.

What can you say about the adoption of response surfaces and what is the level of integration between simulations and velocity prediction programs (VPPs)?

Ubaldo Cella: VPPs provide a crucial framework for the development of sailboat design. They are algorithms that solve the balance of forces and moments acting on a vessel in order to predict yacht performance by modelling the physics of its various components, such as aerodynamic forces on sails, hull hydrodynamics, lift/drag of appendages, and inertial properties. The accuracy of VPP modules is essential to estimate yacht response in various sailing conditions and to determine design quality. However, to integrate VPPs into effective optimization procedures, it is essential to strike the right balance between the accuracy and the computing costs of the analysis techniques used. Implementing analytical formulations and metamodels based on high-fidelity analyses are therefore key areas of study for improving the productivity of design methodologies.

This is because identifying the optimal design parameters requires a significant number of simulations to be performed, and the computing costs can quickly become a bottleneck if not managed effectively. It is therefore crucial to identify the optimal level of accuracy required for each VPP component and to adopt analyses capable of balancing computing cost and accuracy. Achieving this balance is a continuous challenge, and there is ongoing research to improve the accuracy of VPPs while keeping the computing cost within manageable limits [5].

For more information:
Marco Evangelos Biancolini - University of Rome Tor Vergata
biancolini@ing.uniroma2.it

References

[1] rbfLAB (27 May 2022), “DemoFly, sea tests of the sail prototype made by rbfLAB”, www.rbflab.eu [online]. Available at: www.rbflab.eu/demofly-sea-tests-of-the-sail-prototype-made-by-rbflab/

[2] M. Calì, D. Speranza, U. Cella, M. E. Biancolini, “Flying Shape Sails Analysis by Radial Basis Functions Mesh Morphing”, *Design Tools and Methods in Industrial Engineering*, ADM 2019, Lecture Notes in Mechanical Engineering, pp 24–36, Springer, September 2019, doi: 10.1007/978-3-030-31154-4_3.

[3] U. Cella, C. Groth, S. Porziani, A. Clarich, F. Franchini, M.E. Biancolini, “Combining Analytical Models and Mesh Morphing Based Optimization Techniques for the Design of Flying Multihulls Appendages “, *Journal of Sailing Technology*, Volume 6, Issue 01, February 2021, doi: 10.5957/jst/2021.6.1.151. onepetro.org/ JST/ article-pdf/6/01/151/2452843/sname-jst-2021-09.pdf/1

[4] M. E. Biancolini, I. M. Viola, M. Riotte, “Sails trim optimisation using CFD and RBF mesh morphing”, *Computers & Fluids*, Volume 93, pp. 46–60, 2014, doi: 10.1016/j.compfluid.2014.01.007. Available at: www.sciencedirect.com/science/article/abs/pii/S0045793014000140?via%3Dihub

[5] Byrne et al, “Using Machine Learning to Model Yacht Performance”, *Journal of Sailing Technology*, Volume 7, pp. 104–119, 2022, doi: doi.org/10.5957/jst/2022.7.5.104

The iconic barges sailing up and down Europe's rivers could soon be replaced by an electric or hydrogen powered alternative. Image credit: Thib Ault via Unsplash

Ramping up renewables to power the clean future of inland shipping

Hydrogen and battery-powered barges, tugs, container ships, and ferries for inland waterways will help lower emissions from shipping.

By Michael Allen

Compared to other industries, such as agriculture or energy, transport is the only sector with greenhouse gas (GHG) emissions higher now than in 1990. Sending out around 3% of the world's GHG emissions, the maritime industry is a particularly hard nut to crack.

The EU Green Deal is seeking to make Europe climate neutral by 2050. The International Maritime Organization (IMO) is taking measures to cut emissions from shipping by at least half by 2050. Electrification promises to become a key technology to reduce shipping emissions, but maritime battery technology is not yet mature enough to allow this.

"Batteries for maritime industries are not yet at the cost point they should be to make full electric shipping widely possible," says Jeroen Stuyts, of Flanders Make, a research institute in Belgium.

"At the moment," Stuyts says, "some vessel types are more suitable for battery-power adoption than others. These include ferries that run on predetermined routes and smaller container ships sailing fixed routes."

"Their journeys are very predictable and have reliable work patterns. This makes it easy to analyse their energy consumption and work out the costs of electrification, in order to understand whether or not it is economically viable."

Driving down costs

On the other hand, ocean-going container and cargo ships on intercontinental voyages are difficult to electrify because of the variability of conditions. "You need a very, very, very big battery to cope with all the uncertainty," Stuyts explains.

Battery-powered ships are a costly solution "Primarily because the batteries that are developed for maritime use are typically bespoke for one particular ship design," Stuyts says. "This makes them unique, and therefore expensive."

Stuyts is co-ordinator of the EU-funded SEABAT project [1], which is developing an electric battery for the maritime sector. They are working to bring innovation in battery

> Batteries for maritime industries are not yet at the cost point they should be to make full electric shipping widely possible.
>
> Jeroen Stuyts,
> SEABAT

technologies from the automotive industry to the maritime sector and drive down costs.
They aim to achieve this by developing a modular battery system that can be used in a wide variety of ships.

It doesn't help that little research on batteries is focused on the maritime sector. The energy requirements for ships are completely different to other vehicles. Most cars have a battery capacity of around 40–100kWh (kilowatt-hours), but ships require 10, 20 or even 100-times more capacity than that.

Mix and match

The system being developed by SEABAT consists of standardized modules for the various battery components, such as battery cells, cooling systems, and power electronics. These can be combined in racks and scaled to create batteries of different sizes. SEABAT will develop a way to build custom batteries made from standardized components that can be mass-produced, drastically reducing costs.

The modular nature of the system also allows different battery cell types, or even other storage types such as super-capacitors, to be combined. It will be possible to create battery solutions tailored to the specific needs of the ship, Stuyts explains, and to upgrade the system as new battery cell technologies are developed.

Green hydrogen

When you can use batteries, they are a good, energy efficient way to decarbonize, according to Jyrki Mikkola, at VTT in Finland. But we also need solutions for situations that cannot be tackled with batteries alone.
In the EU-funded FLAGSHIPS project [2] which Mikkola is coordinating, project partners are developing hydrogen fuel cells for two commercially-operated demonstration vessels. By designing a barge and a container vessel fuelled by green hydrogen, they hope to launch the era of clean waterborne transport in Europe.

FPS Waal, a Dutch-registered 110-metre inland container ship will run around 80 kilometres back-and-forth from Duisburg in Germany to Rotterdam in the Netherlands. The ship, carrying 200 standard shipping containers, will replace a diesel vessel that currently transports goods between the inland regional port and the international port in Rotterdam.

Seine barge

The other vessel is a 60-metre-long inland navigation barge for cargo transport on the river Seine. Named "Zulu 06", the brand-new vessel brings hydrogen powered navigation to the centre of Paris. Zulu 06 is planned as the first zero-emission hydrogen-propelled river barge in Europe. As Mikkola explains, the river craft will distribute goods along the river Seine in the very centre of Paris, with a view to replacing some freight transport on the city's congested roads. "The barge will have its own crane so it can basically unload anywhere along the river," he says.

The propulsion system that the FLAGSHIPS project developed is also modular. Each hydrogen fuel cell module is 200 kilowatts and multiple modules can be linked together to scale up to as much power as needed. The Paris barge will use two fuels cells, providing 400 kilowatts of power, while the containership FPS Waal will have 1.2 megawatts of power from six fuel cell modules.

Developed in partnership with industry, the zero-emissions barge is due to start operating in 2023. With its urban-logistics route changing depending on the demands of the day, it is a good example of a situation where standard electrification technology could present challenges. Changeable route navigation makes it difficult to build up a network of charging points according to Mikkola. The hybrid hydrogen propulsion system will extend the barge's range and provide more operational freedom.

Hydrogen hybrids

Hydrogen-powered hybrid vessels still employ electric motors driven by batteries. But they enable the use of smaller batteries that are constantly recharged by the hydrogen fuel cells. Hydrogen is relatively energy-dense and light, compared to batteries, allowing for longer ranges with less overall impact on ship design. As well as cutting GHG emissions, hydrogen vessels do not emit pollutant gases like sulphur dioxide and nitrogen oxides.
"Locally produced green hydrogen, combined with fuel cell technology, offers a very interesting and viable option for the shipping industry, who is eagerly looking for solutions to make their operations green in inland waterways," Mikkola says.

Since fuel cells and hydrogen vessels don't have local emissions, air quality in cities could be vastly improved if the technologies can be implemented on a large scale. They could even replace toxic gas-belching trucks and vans, as well as current waterborne transport systems.
The EU is committed to a long-term shift toward clean energy to reduce its reliance on Russian fuels. Shifting to technologies like hydrogen would enable the production of fuel on a local and regional basis.

Local hydrogen

"We can set up small electrolysers in different ports in Europe and produce the (hydrogen) locally," says Mikkola. "We don't need to ship it from somewhere else."

FLAGSHIPS is part of a wider EU-funded strategy to develop hydrogen-powered freight transport for Europe's waterways. The project hopes its two vessels will raise awareness of zero emissions waterborne transport, encouraging others to follow suit.

For Stuyts and SEABAT, it is also important that research on these future technologies is conducted in Europe. Most of the raw materials needed for batteries come from outside Europe, primarily Asia. This means that experts in Europe need to understand how to design and build battery systems. EU funding for research and innovation in this area plays an essential role in helping Europe remain competitive in this market. "Otherwise, we have neither the materials nor the knowledge," says Stuyts.

References

[1] cordis.europa.eu/project/id/963560
[2] cordis.europa.eu/project/id/826215

This article was originally published in *Horizon, the EU Research and Innovation magazine.*

ec.europa.eu/research-and-innovation/en/horizon-magazine/ramping-renewables-power-clean-future-inland-shipping

Rendering of EEMS Traveller cargo ship with two suction sails installed. © bound4blue, 2022

Ships harness wind for voyage to a cleaner future

New sails and boat designs are set to help shipowners slash fuel costs and emissions.

By Gareth Willmer

There is no mistaking Cristina Aleixendri's enthusiasm – and competence – when it comes to talking about how wind-assisted shipping is on the verge of making a planet-changing comeback.

Aleixendri founded a company called bound4blue with two fellow Spaniards in 2014 to develop sail technology inspired by their training in aeronautical engineering.

Dream comes true

"When we started, we were seen as crazy engineers for wanting to bring sails back to ships," she says. "But when we speak to shipowners today, they tell us we'll go back to wind, and it will never be abandoned."

It's easy to understand why. The shipping industry accounts for about 3% of global greenhouse-gas emissions and is trying to move away from heavy fuel oil, which is highly polluting. "Wind-propulsion technology will become a standard," says Aleixendri. "It started as a dream of mine. Now, I see it less as a dream and more of a reality."

Not only has Barcelona-based bound4blue attracted growing interest from shipping firms in its wind-assisted propulsion system but Aleixendri has achieved significant personal recognition for her efforts.

In 2019, she made the Forbes 30 Under 30 list for manufacturing and industry in Europe. The following year, Aleixendri won the European Institute of Innovation and Technology's Woman Innovators Award recognizing inspiring female entrepreneurs.

Wind in the sails

Now, bound4blue is coordinating an EU-funded sails project that borrows the company's name and runs for two years through February 2024. There is big room for growth in wind-assisted shipping. As of September 2022, only 21 large commercial ships globally were equipped with the ability to harness wind energy, according to the International Windship Association. Though predicted to more than double to as many as 50 vessels this year, that is still a drop in the ocean compared with the global fleet.

Wind energy is viable for a variety of vessels, including cargo carriers, tankers, ferries, and cruise ships, according to Aleixendri. "It's a massive market because there are more than 60,000 ships sailing worldwide that could benefit from such solutions," she says. "This is very nascent."

As 2023 dawned, the entry into force of new regulations by the International Maritime Organization on energy efficiency and carbon emissions is also expected to spur growth. "I think it's the right moment to invest in wind propulsion – it's a very sweet spot for us," says Aleixendri, who is her company's chief operating officer and earned a Master of Sciences degree in aerospace engineering from the Polytechnic University of Catalonia.

Suction fan

Bound4blue has developed what's called an autonomous suction-based sail, which looks nothing like a traditional one. It has the appearance of a cylinder-shaped tower that rises from the ship's deck.

Traditional sails work by "catching the wind". The wind creates a higher-pressure area behind the sail compared to its other side. This difference in pressure generates a force known as "lift" that propels the ship forward.

By contrast, bound4blue's "eSAIL" contains a suction fan to draw air inside the tower as wind flows around it, creating stronger lift to power the boat. "This results in six or seven times the lift of a conventional rigid sail and could reduce fuel consumption by up to 40% if combined with better vessel design and adjustments in routes to take advantage of prevailing winds", says Aleixendri.

"The eSAIL is best suited for the types of wind conditions found in the North Atlantic and North Pacific", she says "Though its use is by no means exclusive to those routes."

Emission savings will vary depending on the general wind conditions on different routes. For example, bound4blue estimates that a merchant ship sailing the 25,000 kilometres from southern Brazil to north-eastern China could save 26% on fuel and emissions.

While it is still early days, some first movers have already reported savings of 15%. Bound4blue has also signed a range of deals with shipping firms including Japan's Marubeni and French-owned Louis Dreyfus Armateurs. "We have more demand than we can supply today, so we're very happy about how it's going," says Aleixendri.

"While new technology has previously been seen as risky to install on ships, wind-assisted options like bound4blue's are starting to make economic sense and can pay for themselves in fuel savings within five years," she says. "In the end, wind propulsion is providing free, renewable energy that you don't have to store or invest in infrastructure to supply," says Aleixendri.

Vessel design

Amid the promise of wind-based options, a challenge arises: ensuring they are properly implemented to achieve their full performance potential or preventing negative knock-on effects on how a ship runs. So another EU-funded project, OPTIWISE, is investigating how the overall design of vessels can be adjusted to optimize wind-assisted propulsion.

"Better attuning ships to the technology can help improve sailing efficiency and emission savings," says Rogier Eggers, who leads the three-year project running through May 2025.

Design modifications could also help overcome some of the potential negative consequences of installing sails on ships. Doing so may, for instance, create an obstacle for passing under objects like cranes in ports or even affect ships in such a way that they struggle to stay on course. "That's simply not acceptable, so one has to look at the shape of the hull and appendages such as rudders to make sure that you get the ship in balance," says Eggers, a senior project manager at Dutch maritime research institute, MARIN.

Over the next couple of years, OPTIWISE plans to use scale models of ships several metres in length to test wind systems and the effects of technological improvements in various sea conditions. The project also intends to employ computer-based voyage simulations and machine learning. "Innovations could deliver savings of well over 30% in carbon emissions, maybe even reaching as much as 50%, if effectively delivered," says Eggers.

Blast from the past

If wind technologies can be successfully integrated, methods like suction sails, wing sails, and cylindrical spinning rotor sails being produced by partners in OPTIWISE could gain real traction, he says.

Adopting such rotor sails would resurrect a wind-based technology invented a century ago by Anton Flettner, a German engineer. It failed to become widely adopted as a result of the growing popularity of diesel fuel at the time. "Several suppliers have been pretty active with wind technology and have been getting increased interest from the shipping market for installations," says Eggers. "Before, there was a big reluctance to put such things on ships, but devices like Flettner rotors, suction sails, and wing sails are now gradually being trusted by the industry."

This transition promises to set the maritime sector on a course towards slashing emissions. "We are at the start with shipping in moving towards a zero-emission future," says Eggers. "The number of ships now equipped with wind propulsion is still tiny compared to the world fleet, but the hope is that we will soon be seeing hundreds of ships being equipped per year."

The largest suction sail in the world being installed at a shipyard. © bound4blue, 2021

References

[1] cordis.europa.eu/project/id/190145145

[1] www.forbes.com/under30/list/2019/europe/manufacturing-industry/#56a6839a7d17

[1] eit.europa.eu/news-events/news/changing-mindsets-diversity-matters

[1] rina.org.uk

[1] www.imo.org/en/MediaCentre/HotTopics/Pages/EEXI-CII-FAQ.aspx

[1] cordis.europa.eu/project/id/101056769

This article was originally published in *Horizon, the EU Research and Innovation magazine.*

ec.europa.eu/research-and-innovation/en/horizon-magazine/ships-harness-wind-voyage-cleaner-future

Analysing ship resistance easily with a calculation program

by Kyung-mi Kim
Tae Sung S&E

As the types of ships become more diverse and the shipbuilding period shortens, vessel shapes must be developed considering their fluid-dynamic performance. In general, a ship's resistance is predicted by performing model-based experiments. However, a computational fluid dynamics (CFD) analysis is usually performed at the initial design stage to address cost and environmental issues.

The flowchart in Fig. 1 shows the overall process of analysing ship resistance. CFD analysis generally consists of four steps: 1) geometry, 2) meshing, 3) solver, and 4) visualization of results. The first step in the analysis process is to generate a grid that fits the geometry to be simulated. However, generating the geometric grid to calculate the resistance values for different types of ships requires a significant amount of time. Meshing also requires a high level of experience and technical proficiency.

There is therefore growing demand for numerical simulations that reduce repetitive tasks and can be easily used by users unfamiliar with CFD. To this end, Tae Sung has developed a program based on Ansys SpaceClaim and Ansys Fluent to automatically generate grids and perform simulations for different types of ships.

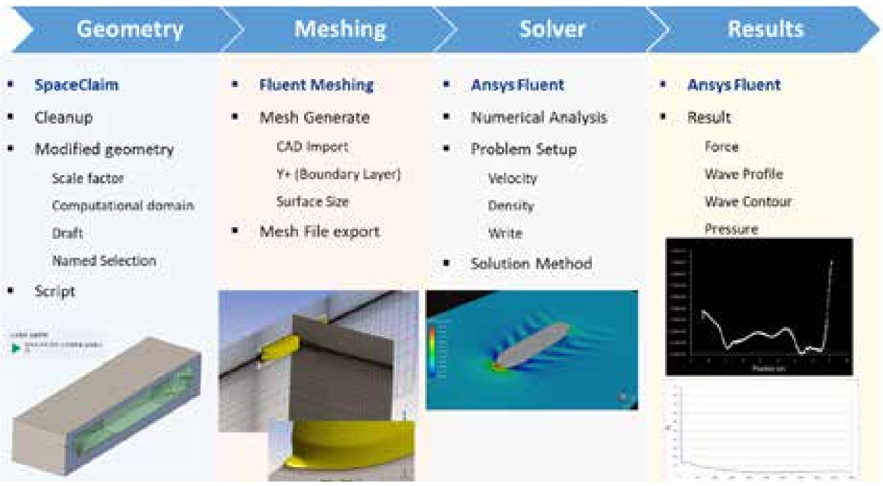


Fig.1. Procedure flowchart.

TSNE's calculation program is developed specifically for ship models and has a fixed size for the calculation domain. There are several prerequisites for using the program.

Prerequisites and constraints

Geometry

Ansys SpaceClaim uses scripting, which sets the scale of the draft, the domain size, etc. to model the geometry. The script can also model the flow domain using imported CAD files (*.igs), however depending on the file, this may require some clean-up by the user before automated modelling to avoid the generation of errors.

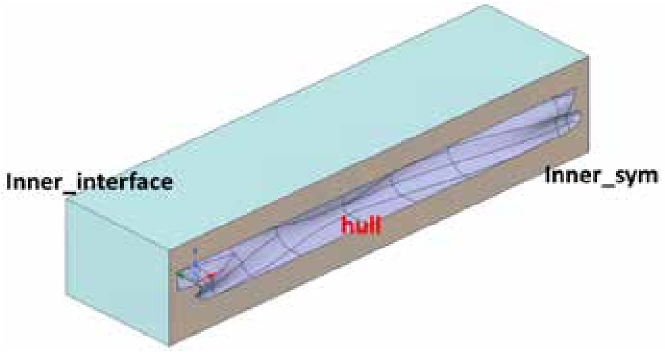


Fig. 2. Named selection.

Once the modelling is complete, the geometry is defined using a named selection set that is saved with the SpaceClaim Direct Modeler (SCDM) file extension (*.scdoc). Only the region around the ship's hull must be named: the ship hull is named "hull", the sides as "inner_sym", and the rest as "inner_interface".

Meshing

After considering the efficiency of the mesh and the accuracy of the simulation, the grid is generated by dividing the flow domain into two parts: the area around the ship's hull, and the ocean domain.

The ocean domain is represented by a hexagonal mesh with the mesh density centred near a free surface. This mesh is saved in a unique mesh file (single_outer_fluid_domain.msh) and locked in the calculation program.
The area around the hull is represented by volume meshes that follow the shape of the ship. They consist of unstructured (tetrahedron) meshes with the boundary layer set along the Y+ axis to generate the meshes more densely around the hull. After creating the volume mesh, convert the type of mesh from tetrahedral to polyhedral in Ansys Fluent to reduce the number of cells. Since the area around the hull and area of the ocean are saved as separate grids, the boundaries between them are treated as interfaces.

Solver/results

Table 1 shows the boundary conditions applied to the simulation.

Model	Volume of Fluid (water and air)	Volume of Fluid (water and air)
Viscous Model	Realizable k-e, Standard Wall Functions	Realizable k-e, Standard Wall Functions
Boundary Conditions	Inlet (pressure inlet)	Velocity (open channel)
	Outlet (pressure outlet)	Atmospheric Pressure (open channel)
	Wall	Non-slip
	Symmetry	
Solution Initialization		Standard Initialization
Etc.		Pressure-based Solver Double Precision Steady (pseudo)

Table 1. Boundary conditions.

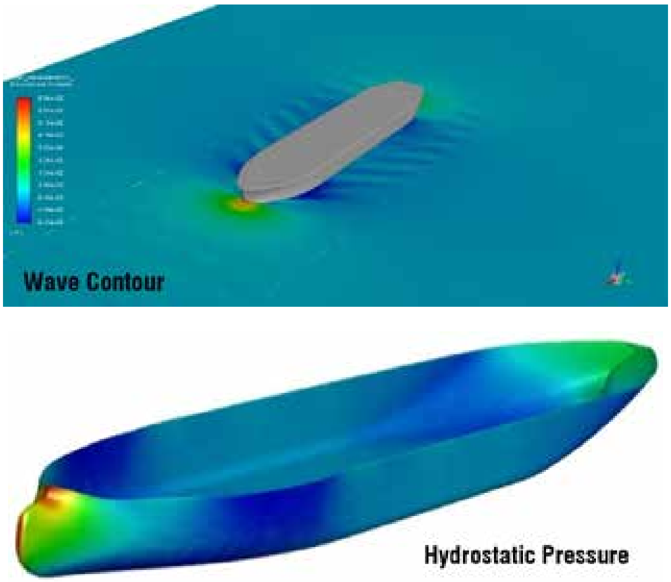


Fig. 3. Results.

Items to confirm when viewing the results include hull pressure, wave contours around the hull, etc. The items to be analysed and the viewpoints are pre-created as *.scm and *.vw files so that they can be applied immediately.

Configuring the Ship Resistance Analysis program

The hull calculation program uses Ansys SpaceClaim to write the script; Fluent with Journal for the calculations; and Visual Basic for the windows-based graphical user interface (GUI). The program consists of three steps: mesh generation, solver setup, and result visualization (Fig. 4).

The program requires the following files to be saved in the same folder: the hull geometry (*.scdoc); the mesh file of the ocean

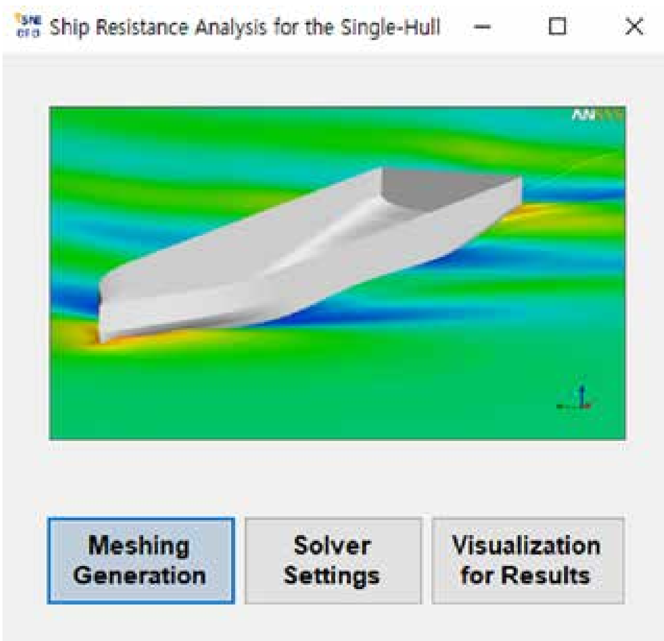


Fig. 4. Ship Resistance Analysis program GUI.

domain (single_outer_fluid_domain.msh); the calculation program (ship resistance.exe); and the SCDM perspective and diagram file for post-processing.

After running the ship resistance.exe file, the user clicks on Meshing Generation which opens the Fluent Meshing window and moves the user from generating the mesh to saving it.

A work path is selected in the Working Directory, and the SCDM file in which the named selection has been defined is selected in the CAD Files (SCDM Files). Values for Min., Max. and Growth Rate are entered in the Surface Mesh Sizes field to set the size of the surface mesh. Volume meshes are generated based on the size of a surface mesh.

The user sets the conditions for creating the boundary layer near the hull in the Boundary Layer Settings. The First Height field sets the size of the first cell along the Y+ axis, and then the user enters the values for the number of layers and growth rate.

The Create Boundary Box defines the domain for the denser meshes. The values of the X and Y directions are fixed in the domain, so the user only sets the values for the Z axis. The values for Z Max. and Z Min. determine where the mesh is densely applied near the free surface, and the value for Grid Size in the BOI (body of influence) Box determines the size of the free surface mesh for the water.

After entering all the values, clicking Generate automatically creates the mesh and saves it as “hull.msh”. The program can then be closed.

Clicking on Ansys Fluent switches the program to Fluent Solve mode. The Solver Settings button initiates the second step of the Ship Resistance Analysis program where the user inserts the minimum conditions for the simulation in the Ansys Fluent dialogue box.

In such window, the user sets a path for the Working Directory and selects the “hull.msh” file created in the previous step for the Mesh Files (*.msh). Under Options, the user can select Double Precision to increase the accuracy of the simulation, or Read Case for meshes created with a non-Ansys product. The user-entered parameters: velocity, density, and area are used to determine the drag coefficient. Entering the X, Y, Z coordinates into Wake Plane at Propeller box generates a plane for determining the wake distribution.

After inserting the values, the user clicks the “Solver Settings” button to save the Case and Data file with the conditions set. The software then performs the calculation with sufficient iterations. Once complete, clicking on the “Visualization for Results” button initiates the final step of the Ship Resistance Analysis program.

The Visualization for Results window permits the user to select a working directory; the Case and Data Files to read; the version; and to select the results to be included in that working directory.

Conclusion

This Ansys-based calculation program makes the analysis of ship resistance more accessible. The GUI assists users who are unfamiliar with mesh creation or who want to spend less time on repetitive tasks, allowing them to obtain the best results with minimal input and make their analyses more efficient.

For more information:
Kyung-mi Kim – Tae Sung S&E
kmkim@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. The company 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, defence, chemical industries, etc.) and is expanding its business scope to research innovative technologies and apply them in the field. It strives to become a global engineering company and increase its potential as a sustainable engineering company. Tae Sung S&E partners all engineers who endeavour to solve challenges. Tae Sung S&E will work with you to achieve “NO PROBLEM, BE HAPPY”.

Exploring multi-resolution particle CFD methods

by James Crist
EnginSoft USA

Particle-based CFD (computational fluid dynamics) methods have become very popular in recent years due to the simplicity of the model configuration process and the ability to solve free surface problems such as splashing. One of the methods for free-surface problems is called moving particle simulation (MPS) and is frequently used in the automotive industry to evaluate splash patterns and churning losses in gearboxes.

The method assumes incompressibility (constant density) of the fluid which allows it to use larger time steps than most other particle methods. Historically the disadvantage of particle methods is related to their ease of use: the end user is able to preprocess the model by specifying only a fixed particle size that the software uses to discretize the model, thus providing a constant spatial resolution of the domain.

This particle size must be set according to the smallest feature, so large models with small features require a large number of particles. As the number of particles increases, additional memory and computing power are required which can make problems impractical to solve. Several approaches have been developed to manage this problem, each with its own advantages and disadvantages.

The fundamental challenge is that particle methods can guarantee conservation of mass and momentum by virtue of resolving fluid with discrete, persistent particles, but this guarantee must be relaxed to implement multiple particle sizes in a single model.

The MPS approach allows one or more regions to be defined in which the particle size is a fraction of the global size. Around these regions a transition zone is created in which the particles can split (or merge) so that mass and momentum are conserved.

This paper will discuss the challenges inherent in variable-resolution particle methods, some existing methods, and the new approach taken by MPS.

History

Particle CFD methods were developed to solve a class of problems difficult for traditional mesh-based codes. Smooth particle hydrodynamics (SPH) was initially developed for the astrophysics community to analyse the formation of astronomical objects such as stars or solar systems. In these models, density varies from a vacuum to the interior of a star, so SPH particles are constructed to manage large density gradients. An SPH particle is a distribution of mass centred around a point. To solve for the system's mass distribution, one simply takes a superposition of all particle mass distributions [1].

MPS was introduced by Kashizuka and Oka as a faster particle method for purely incompressible liquid flows [2]. It describes a particle as a sphere of fluid where particles are not allowed to overlap. This approach has found success in modelling free-surface problems such as gearbox oil splashing where meshing is a time-consuming process.

In both cases these methods discretize a problem into an array of fixed-size particles and then allow the particles to move based on their interactions with their neighbours. Moving boundaries are easily represented by special wall nodes which interact with fluid particles. Motion can be applied to these wall nodes without the need for complex moving or deformable meshes since a particle is not required to exist on the surface of the wall.

Conservation of energy

Unlike the Eulerian approach which has fixed connections between nodes over many time steps, particle CFD methods dynamically evaluate the interactions between a particle and its neighbours at each time step. At each time step a particle will identify its neighbours as particles within a certain distance, or within a local sphere of influence. This sphere can be described as a ratio of particle diameter which introduces the first challenge when implementing a particle solver with multiple particle diameters: the sphere will change with particle size.

If one chooses to make the sphere of influence constant, relative to particle size, then smaller particles will have a smaller sphere of influence. Close to the boundary of a high-resolution region there will be pairs of large and small particles where the small particle is in the sphere of influence of the large particle but not vice versa, leading to unbalanced forces and ultimately to the creation or removal of energy.

If one chooses to make the sphere of influence constant, regardless of particle size, then smaller particles will have more neighbours than larger particles. As the number of neighbours increases, more calculations are required to evaluate the net force on a given

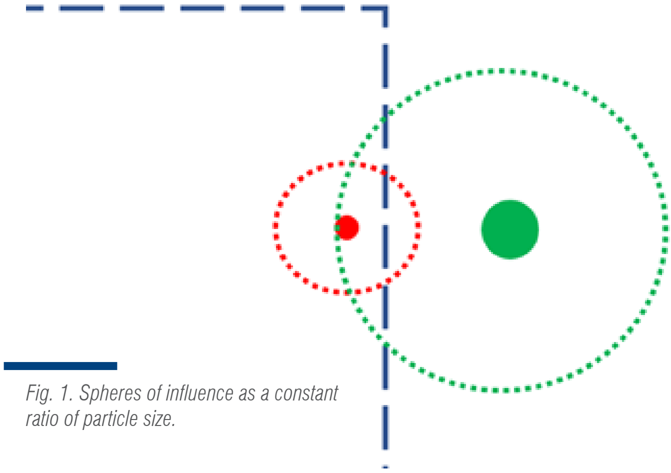


Fig. 1. Spheres of influence as a constant ratio of particle size.

particle, increasing the computational cost. In this case a multi-resolution model can have more total neighbours than the same model with entirely smaller particles if the ratio of small-to-large particles is sufficiently high.

The MPS multi-resolution implementation adopts a hybrid approach by introducing a transition region surrounding the region of smaller particles. Within the transition region the small and large particle problems are separated, with the properties of the large particles acting as boundary conditions for the small-particle problem. The properties of small particles are smoothed and applied in a similar way to large particles [3].

Conservation of mass

The next challenge that arises when implementing a multi-resolution method is conservation of mass. When defining a multi-resolution model it is necessary to specify certain regions that can contain smaller particles. As the simulation evolves particles will cross the boundary of this region and must be converted from small to large or vice versa.

Clearly this cannot be done one for one particle while conserving mass. Instead, some fixed ratio of large and small particles are exchanged. Even if this ratio is an integer number of small particles per single large particle it may not be possible to conserve mass. A problem arises when a single particle, or an excess particle, leaves the high-resolution region in a given time step the solver must relax mass conservation or the high-resolution boundary.

The MPS implementation chooses to do the latter by allowing small particles to exist in the transition region described above. In the time it takes for a particle to travel the length of the transition region, it becomes more likely to have enough neighbours leaving the region to constitute a large particle [3].

Conservation of momentum

The case of a single particle leaving a high-resolution region can be extended to highlight another challenge with variable-resolution particle methods. When a group of small particles is replaced by a large particle, multiple momentum vectors must

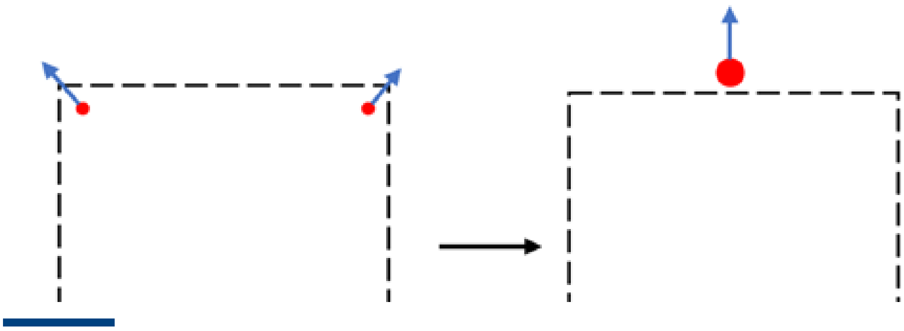


Fig. 2. Merging particles with opposing velocity vectors.

be replaced by a single vector. For a group of neighbouring particles, they are likely to have similar velocity vectors but this will not always be the case. If a small number of particles leave the high-resolution region, the resulting velocity vector may lead to non-physical behaviour.

The MPS implementation limits this behaviour by interpolating the velocity and other properties of the deleted particles to a neighbouring point representing a large particle. This interpolation is distance-weighted so a relatively distant particle will have little impact on the direction of travel of the larger particle.

Time steps

Both SPH and MPS have stability criteria for time steps based on particle size, with a typical rule for MPS being

$$\Delta t = C \frac{l_0}{u_{max}}$$

where C is the Courant number, l_0 is the particle diameter, and u_{max} is the maximum velocity in the system. It is clear from this equation that small particles require a proportionally smaller time step than large particles.

Lowering the time step of all particles based on the minimum size would result in unnecessary calculations so the MPS implementation introduces additional time steps.

The time step for the smaller particles is reduced by the same factor as the particle size resulting in multiple sub-steps of small particles between each full-time step of large and small particles. Since the sub-steps can result in unstable configurations of overlapping particles in the transition region an additional step is added. Particle displacement is applied to the overlapping particles to move them slightly into a more stable configuration [3].

Conclusion

While particle methods have many advantages over their mesh-based counterparts their strengths can become weaknesses. The trade-offs that allow them to efficiently simulate high density gradients and sparsely populated domains make changing the resolution across the domain challenging.

The MPS approach involves decoupling large and small particle regions with a transition zone between them. Within the small-particle regions the time step is reduced locally to preserve stability without slowing down the rest of the model.

For more information:
James Crist - EnginSoft USA
james@enginsoftusa.com

References

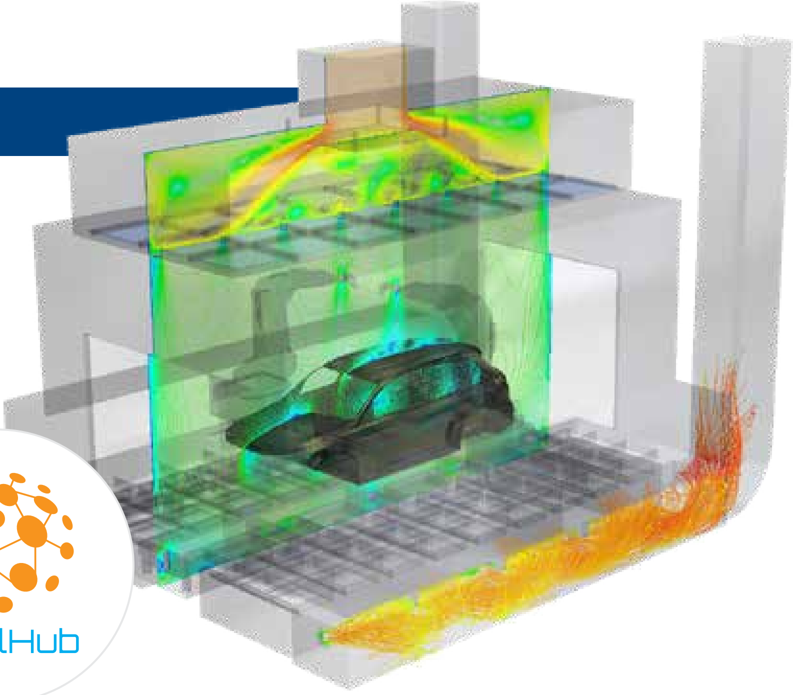
[1] EnginSoft USA, "Smoothed Particle Hydrodynamics" [Online]. Available: enginsoftusa.com/Smoothed-Particle-Hydrodynamics.html, accessed 15 January 2023.

[2] S. Koshizuka and Y. Oka, "Moving-Particle Semi-Implicit Method for Fragmentation of Incompressible Fluid," *Nuclear Science and Engineering*, vol. 123, no. 3, pp. 421–434, Jul. 1996, doi: doi.org/10.13182/nse96-a24205.

[3] *Particleworks 8.0 Theory Manual*, Prometech Software Inc., 2023

Using CFD simulation to optimize the ventilation system of a spray booth

by Luca Zanellato
SimulHub



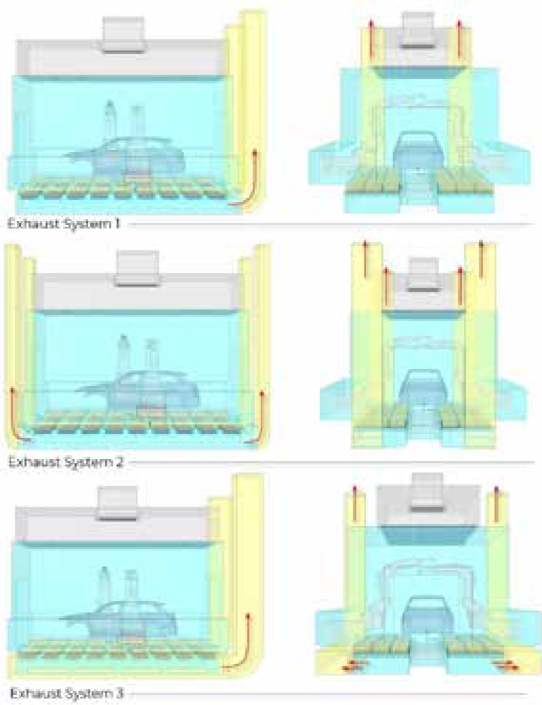
Overspray is a key problem in the paint application process. This phenomenon is caused by inefficiencies of the robots inside the spray booth and can best be solved by balancing the air flow properly in the paint application area.

Ventilation airflow serves two purposes:

- It removes overspray using sufficient down draft;
- It forces the air through a filtration system (known as a scrubber) that separates the paint droplets from the air.

This paper discusses and compares three spray booths which use different exhaust systems. The main objectives of this analysis are to:

- Optimize the exhaust ducts to achieve an even distribution of the air flow while respecting the customer's space constraints;
- Calculate the pressure drop of each design.



1) Two exhaust ducts connect directly to the base of the scrubber.

2) Four exhaust ducts connect directly to the base of the scrubber.

3) Two exhaust ducts connect to two additional ducts located under the robot.

Fig. 2. Three different exhaust system scenarios.

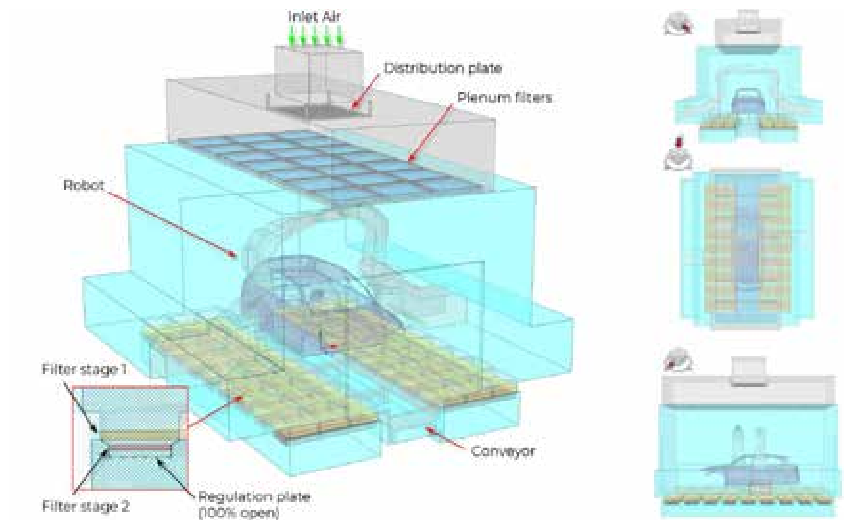


Fig. 1. Geometry of the spray booth.

The simulation was performed with Ansys CFX, a general-purpose CFD (computational fluid dynamics) software. CFD simulation offers the benefit of predicting the function of the proposed ventilation system before it is produced and installed, thus avoiding post-installation modifications and costs to resolve problems.

System description

There are numerous components in the spray booth (Fig.1):

- Ventilation intake duct;
- Air distribution plenum with filters;

- Spray booth including robots, car body and conveyor;
- Scrubber system with two filtration stages;
- Exhaust ducts in three different configurations as shown in Fig.2.

The simulation geometry is comprised of the air volumes while the solid parts are described as boundary walls. Irrelevant details are simplified in order to focus the simulation on the distribution of the air flow and on the pressure losses.

The fluid is air, modelled as a gas with constant properties (at 25°C) and turbulence is modelled using the SST (Shear Stress Transport) model.

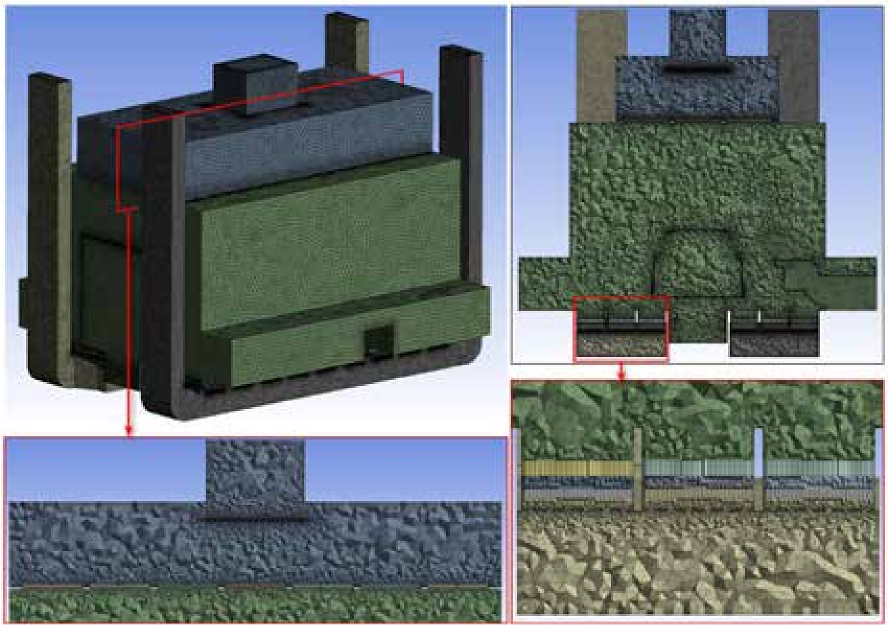


Fig. 4. Mesh composed of 28 million elements.

Name	Boundary	Q	V	P
-	-	m³/h	m/s	atm
Intake duct	Intake velocity	55,500	8.03	-
Outlet duct scenario 1	Outlet velocity	55,500	8.76	-
Outlet duct scenario 2	Outlet velocity	55,500	8.76	-
Outlet duct scenario 3	Outlet velocity	55,500	8.56	1
Booth access gate	Opening pressure	-	-	1

Table 1. Boundary conditions.

Name	DP	Thickness	Q	A	V	K linear
	Pa	mm	m³/h	m²	m/s	kg/m³/s
Plenum	100	50	2,643	1.00	0.73	2,750
Filter stage 1	27	80	1,028	0.46	0.63	550
Filter stage 2	25	50	1,028	0.36	0.79	630

Table 2. Setup of porous domains

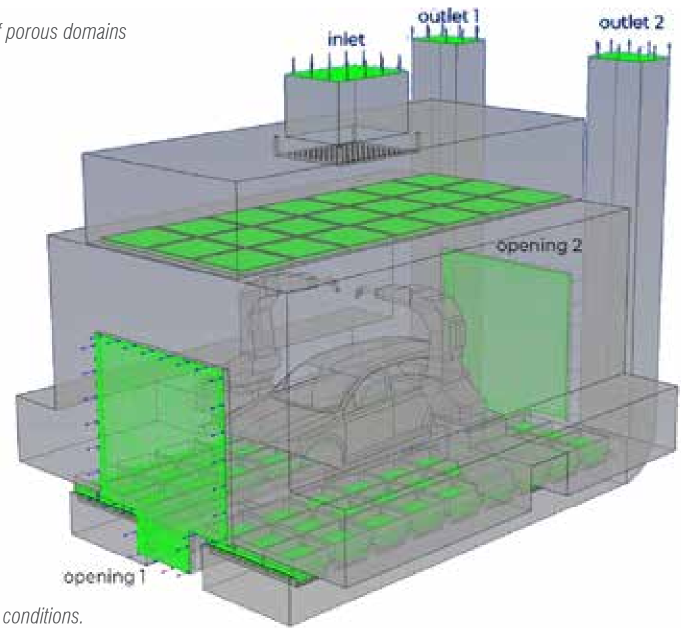


Fig. 3. Boundary conditions.

The filters are modelled as porous domains using porosity and the linear pressure loss coefficient. The boundary conditions are reported in Table 1 and Table 2.

Thereafter, the simulation volume was divided into computational cells using the dedicated mesh-generation tools from the Ansys simulation suite (Fig.4).

Results

To evaluate the downdraft inside the paint application area, the flow rate needs to be checked at the following different heights inside the booth:

- Near the plenum at the top of the booth, air flow must be uniform;
- At a medium height in the booth the flow should accelerate due to the presence of the car body;
- At floor level the air flow over the filters should be evenly distributed.

Scenario 1

The results show that the positioning of the exhaust ducts has a strong influence on the flow distribution inside the booth. The location of the extraction ducts obviously affects the flow distribution over the filters.

Even though this configuration is the most cost effective, the flow is not uniform, and some filters are forced to process more air than their maximum operational limit.

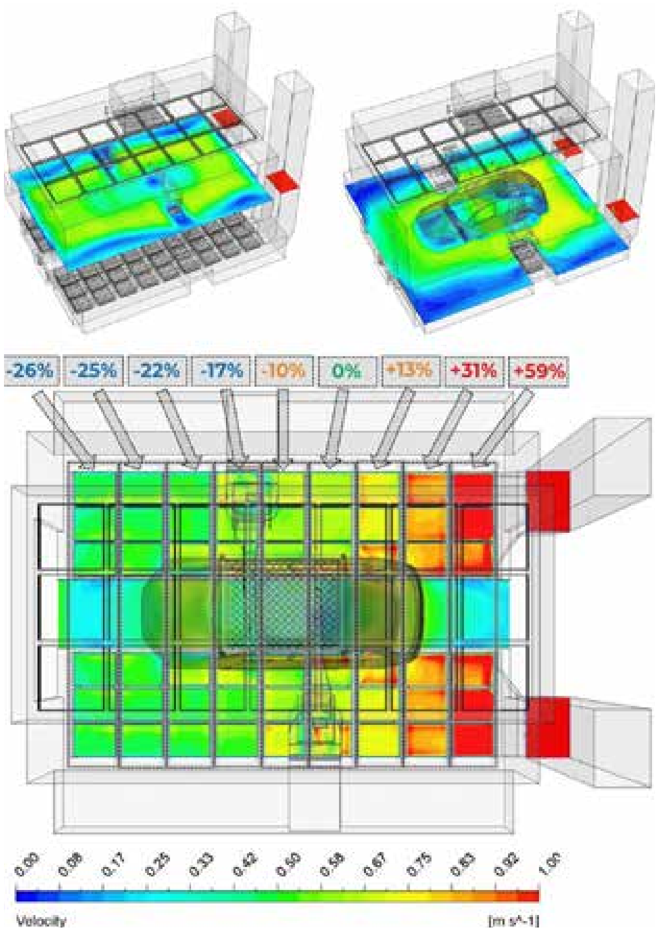


Fig. 5. Velocity distribution for scenario 1.

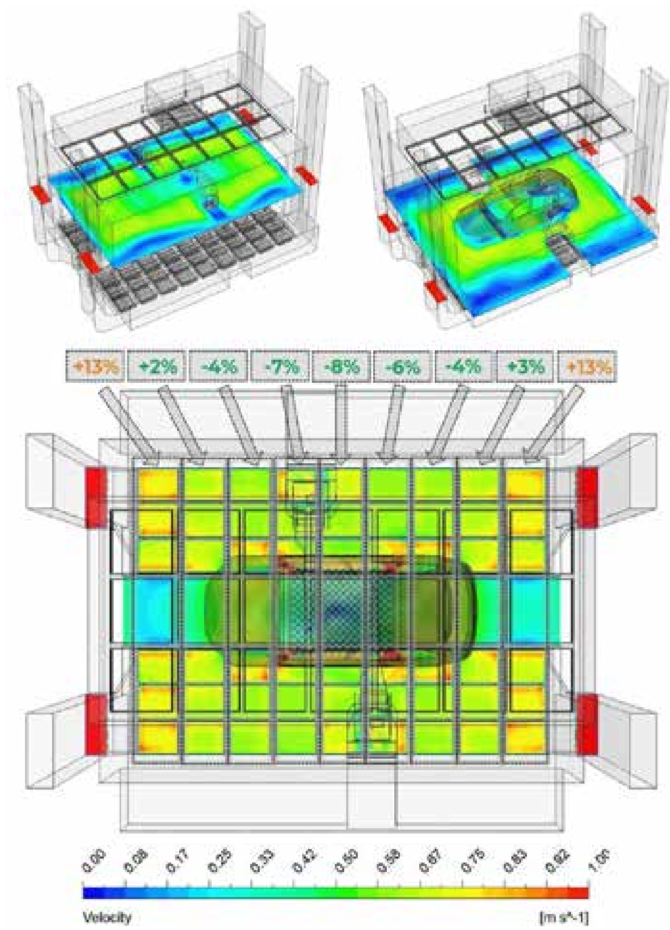


Fig. 6. Velocity distribution for scenario 2.

In these conditions, filters near the exhaust ducts will have a higher load and will consequently be less efficient and become dirty much faster than all other filters.

Scenario 2

As expected, the second scenario has better ventilation. The symmetrical duct design improves the uniformity of the downdraft. The sole concern is that the filters near the exhaust ducts in this scenario are still subjected to a higher load..

Scenario 3

The addition of a second duct below the robots offers the best results, providing optimal air flow distribution. However, this design solution comes at a higher cost (more ducts to install).

The figure shows optimal flow distribution with increased flow under the car body where maximum overspray occurs, thus reducing the risk of the overspray spreading inside the spray booth.

Pressure loss

The following figure graphically represents the pressure losses. They demonstrate that most of the losses are caused by the plenum and the extra loss caused by the additional duct in Scenario 3's extraction system has no significant effect, therefore making the last solution the preferred one from a functional point of view.

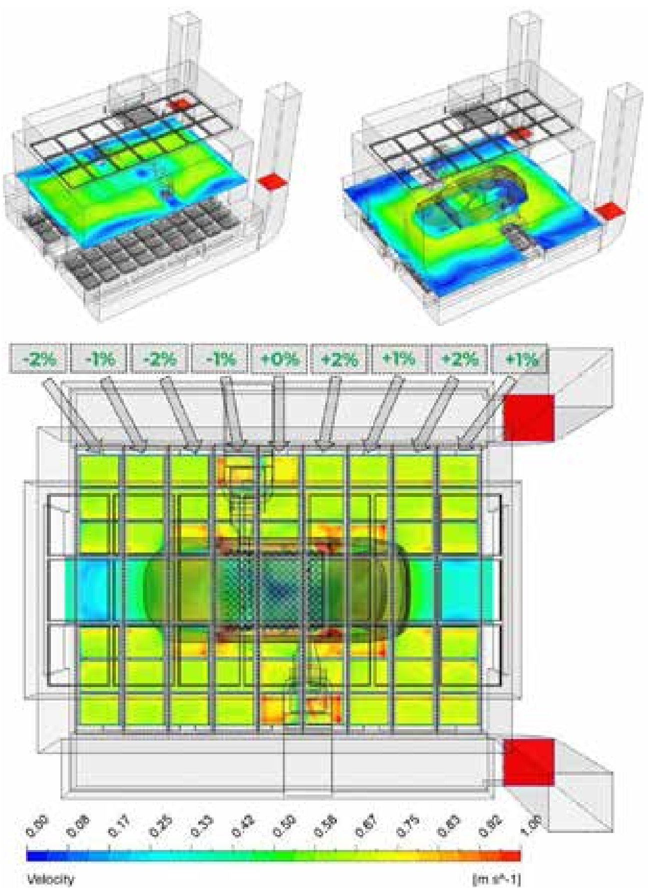


Fig. 7. Velocity distribution for scenario 3.

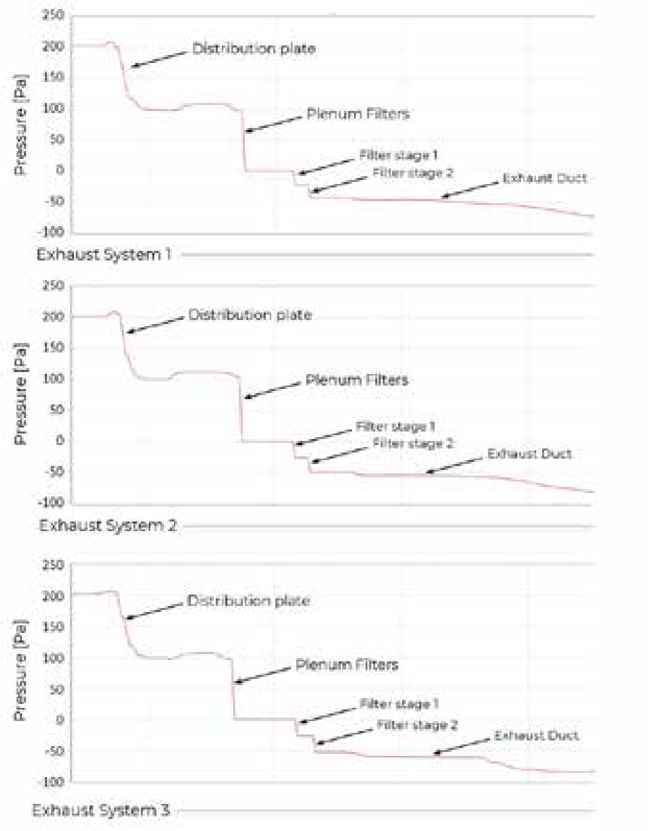


Fig. 8. Pressure loss graphs.

Conclusions

In this paper, we analyzed three different configuration scenarios for the ventilation system of a spray booth.

All three-show similar pressure drops; however, the first scenario has a non-uniform downdraft caused by the position of the extraction ducts. The second scenario has a better airflow distribution during extraction but costs more due to the addition of two further extraction ducts. The final scenario is the best one with the most uniform downdraft.

CFD simulation made it possible to find the best design before the booth was built, thereby improving system efficiency and reducing costs.

About SimulHub

Geico's SimulHub business unit plans, executes, and analyses engineering simulations using professional software in order to predict, prevent, and optimize the operation of production equipment and products throughout the production process.
Thanks to its young and highly experienced team of engineers, SimulHub offers CFD, DES (discrete event) and 1D (one-dimensional) simulations accompanied by the valuable expertise gained in the highly competitive and innovative automotive market. SimulHub places its customers at the centre of its existence, adopting their needs as its own and creating tailor-made, sustainable solutions to meet these needs.

For more information:
Luca Zanellato - Simulation Expert
info@simulhub.com

Integrating multi-camera DIC at MBDA

by Floriane Soulas
EikoSim

As part of its growing expertise in advanced measurement techniques, MBDA has started testing and internal auditing of multi-camera DIC (Digital Image Correlation) measurement resources. After testing these measurement tools on coupon-type test campaigns, the company has moved onto testing this technology on more complex structures using multi-camera systems.

MBDA conducted a compression loading test on a composite material part known as a “hood”. The company identified two distinct zones of interest for this test. The first is located on the tongue of the cap, and the second on the outer face of the arms, called zone 1 and zone 2 respectively (see Fig. 1). Special tools hold the hood in a precise position.

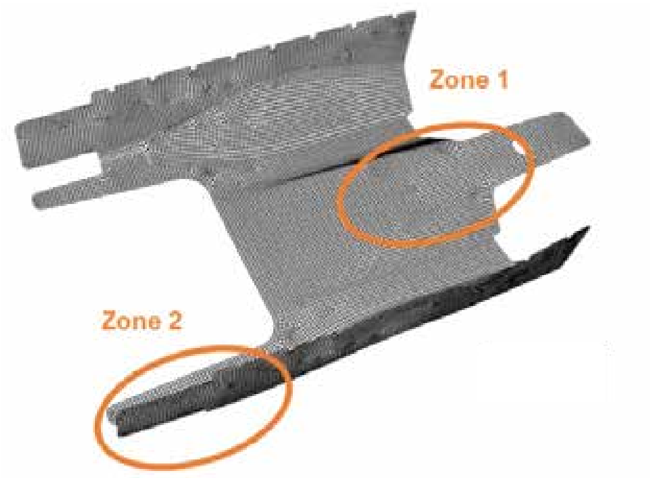


Fig. 1. The two areas of interest on which measurements will be taken using stereo correlation of digital images.

The aim of this test is to validate digital image correlation on a structural compressive loading test, using two camera systems to monitor several zones of interest, and to compare the results obtained with the numerical simulations conducted beforehand. A second interest of this test is to study the use of the measurement technique in a crowded test environment. This test was also an opportunity to train the MBDA teams in good test practices related to multi-camera DIC and in the use of EikoTwin DIC software. All data processing on EikoTwin DIC in this study was carried out by the MBDA teams with the support of EikoSim.

Displacement tracking by multi-camera (or multi-view) DIC

Pre-study

Since the test machine on which the hood is being tested is located in a relatively crowded laboratory, and the hood itself is encased in special equipment, there is limited space in which to operate and place the physical sensors (linear variable differential transformer (LVDT), strain gauges). These restrictions also reduce the available field of view for the cameras, and the space to position them around the part. To address this problem, EikoSim carried out a virtual pre-study of the test using the EikoTwin Virtual tool. This tool is used to virtualize the test scene in order to choose the position of the cameras before the compressive loading test for the two areas of interest (Fig. 2).

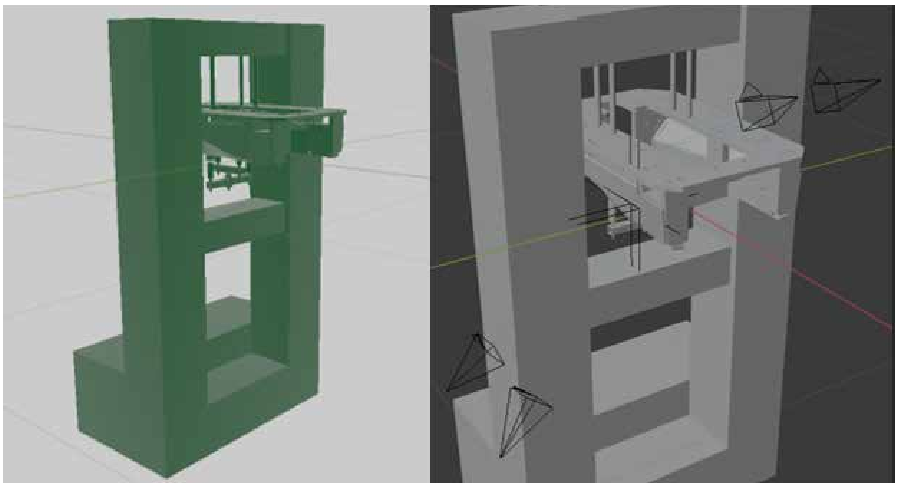


Fig. 2. Positioning of camera systems with EikoTwin Virtual.

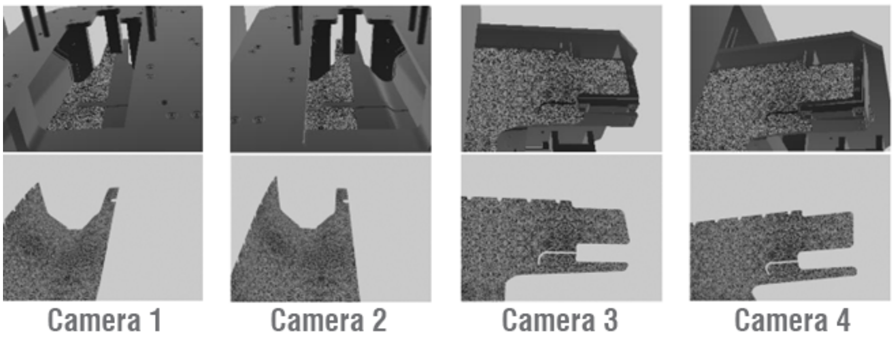


Fig. 3. Virtual images taken during the pre-study.

Using EikoTwin Virtual, it is possible to predict the positioning of the cameras, and to determine the appropriate size of the speckle pattern in relation to the test specifics (taking into account the characteristics of the cameras, the distance between the cameras and the hood, and the refinement of the mesh). The software provides virtual images that are as close as possible to those that will be taken for the real test (Fig. 3).

Measurement	Uncertainty (μm)
Average according to X	5.8
Average according to Y	9
Average according to Z	9.8

Table 1. Average uncertainty obtained from the pre-study.

The virtualization uses the virtual images from the test to determine the minimal uncertainty values that can be expected for the displacement measurement in this configuration.

This represent the measurement noise of the test, calculated under ideal conditions and below which no displacement can be measured. Table 1 shows the minimum measurement uncertainty expected for this test for both areas of interest. This data can be used to predict whether the measurement accuracy expected for a given test is compatible with the anticipated displacement.

Fig. 4. Test configuration with a multi-camera system.

Performing the test

Based on the test configuration defined during the pre-study, we positioned two pairs of static cameras (1 image/second, 4112×3008 pixels) to observe zones 1 and 2, respectively.

Images of both areas of interest were taken simultaneously during the compressive loading test of the hood using the EikoTwin Vision image acquisition software and the multi-camera DIC system (see Fig. 5). MBDA performed several types of pressure loading on the hood.

Fig. 5. Images taken during the test, for the two zones of interest.

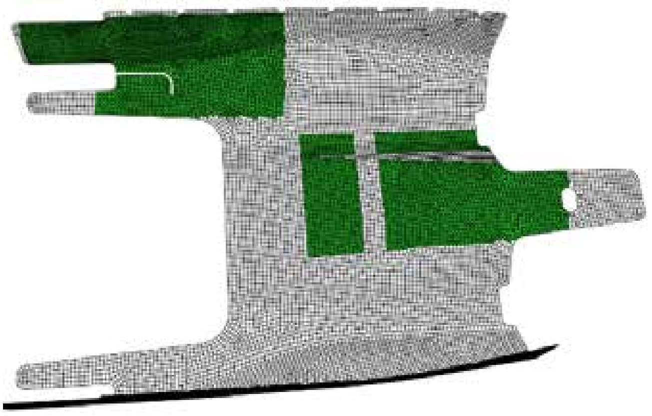


Fig. 6. Definition of the zones of interest directly on the finite element mesh.

Results and outlook

As the finite element model of the structure is extremely complex and refined, only the elements of interest are selected for measurement. Displacements will be measured only in these two zones (shown in green in Fig. 6), and a comparison will be made between the test and simulation results.

The EikoTwin DIC software uses the test images to measure the displacement field directly at the nodes of the finite element model supplied by MBDA, for the two areas of interest. The results of the image correlation can then be compared with the predictions of the finite element model and with any physical sensors present during the test, directly on the mesh.

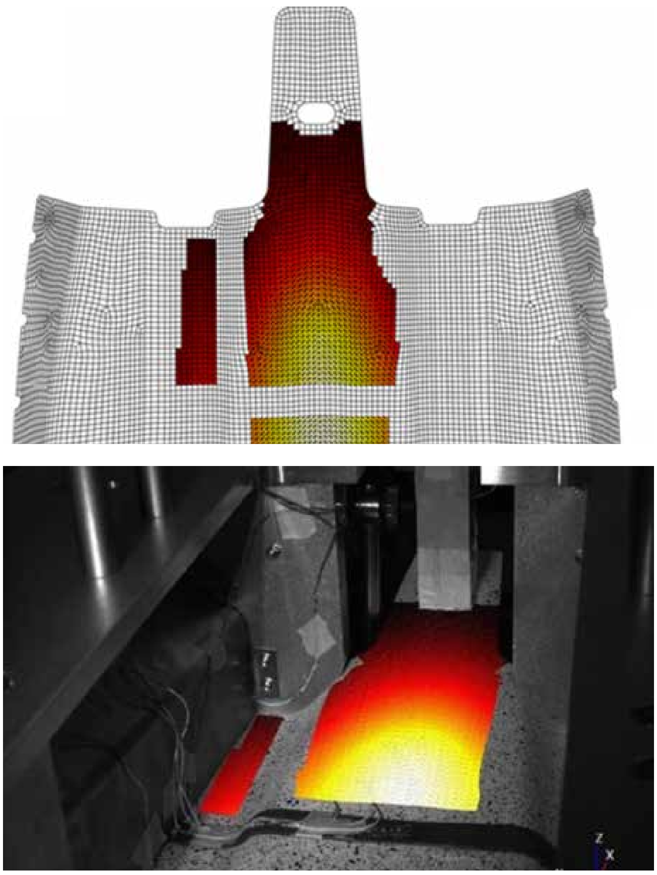


Fig. 7. Normal displacement fields for image correlation (left) and numerical simulation (right) for zone 1.

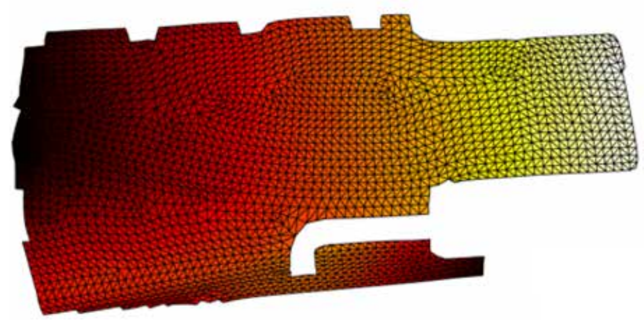


Fig. 8. Z-displacement fields for zone 2.

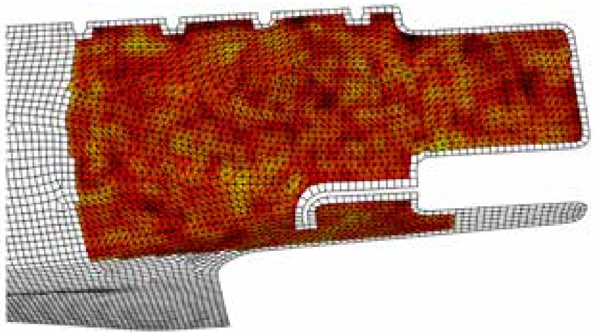


Fig. 9. Difference field between the measurement and the simulation, expressed directly on the finite element mesh.

Fig. 7 shows a comparison of the displacement fields (along the surface normal) obtained between the image correlation (left) and the simulation field projected onto the part for study zone 1 (right). The two fields obtained are very similar. The measured field appears to be more symmetrical in the distribution of displacements around the centre of the part, while the simulation predicted stronger displacements that are more spread out from the centre towards the hood tab.

Fig. 8 shows the measured displacement field for zone 2 along the main load axis Z. EikoTwin DIC also allows the simulation and measurement to be compared directly with the difference fields which show the deviation between the measured and simulated field in the area of interest.

This result for zone 2 is shown in Fig. 9. It is interesting to note that the difference field here is very noisy, indicating that there are no significant differences between the two fields, nor is there any displacement pattern to be found in one of the two fields and not in the other. If MBDA found differences between the sensor measurements, the multi-view DIC, and the simulation, these were in no way significant. In this case, the displacement fields will enable the simulation to be calibrated efficiently, accurately and more extensively than is possible with physical sensors.

Conclusion

MBDA has invested in the acquisition of cameras to generalize the use of multi-view DIC in its range of measuring equipment. EikoSim helped MBDA's test department and design office to improve their skills to facilitate the integration of DIC into the company's process. To do this the DIC tools (hardware and software) were tested on different tests of increasing complexity, from samples and test pieces to a complete

EikoTwin DIC

EikoTwin DIC is an advanced digital image correlation (DIC) software program and system designed to bridge the gap between measurements and numerical simulations. This cutting-edge software employs patented Digital Image Correlation technology to facilitate automatic test-simulation comparisons, making it a valuable tool for various applications.

One of EikoTwin DIC's primary functions is to efficiently detect model errors and pinpoint the sources of these errors. It stands out as the sole global finite element-based digital image correlation software package specifically crafted with simulation validation in mind.

The principle behind EikoTwin DIC revolves around image processing using finite element mesh. It offers automatic calibration centered around the mesh and supports multi-view setups with no technical limitations on the number of cameras employed. This software yields numerous directly accessible fields, including displacements andstrains on the surface of the object under examination.

Furthermore, EikoTwin DIC offers a range of features, such as virtual sensors for tracking displacement and strain gauges. It facilitates immediate test/simulation comparisons based on finite element simulation results, streamlining the process of validating simulations from real-world data.

The software serves as a tool for researchers and professionals, enabling them to connect measurements and simulations, identify and resolve errors, and obtain valuable insights from digital image correlation data.

structure tested with a multi-camera system during a compressive loading test.

A pre-study using EikoTwin Virtual software was carried out to prepare for the test, the positioning of the cameras, and the creation of the speckle pattern. This pre-study was also used to determine the average uncertainty of the measurement. Test data was collected on two separate areas of the hood, which served as the study structure, using a multi-camera system and EikoTwin Vision software. The images obtained were processed internally by MBDA via EikoTwin DIC, after the teams had been trained.

These tests have been encouraging, providing qualitative and quantitative results in areas of study to which only a few physical sensors had previously been applied, making it possible to improve and optimize the numerical model. Today, MBDA is using the multi-camera DIC measurement method and applying it to various in-house test campaigns to validate the characterization of materials or tests on mechanical structures.

About EikoSim

EikoSim is a software company that enables users to leverage validated simulation models to support design decisions. The company supports managers of engineers in charge of structural simulations. It assists its customers to explain discrepancies between tests and models so that they can respond more quickly to program requests and reduce delivery times to the end customer. The EikoTwin software solution applies image analysis and simulation model management to improve both simulations and development cycles.

About MBDA

MBDA is the first truly integrated defence company in Europe and the only European one able to provide missiles and missile systems for each branch of the armed forces, whether in the air, at sea or on land. A joint venture between the three European leaders in aerospace and defence: Airbus (37.5%), BAE Systems (37.5%) and Leonardo (25%). It is a multi-national group with more than 14,000 employees across France, Germany, Italy, Spain and the United Kingdom, as well in the USA.

For more information:
Floriane Soulas - EikoSim
contact@eikosim.com

Earthquake evaluation of a stacker crane deployed in the Mexican Republic

The intralogistics sector uses stacker cranes to transport stacked pallets between racks.

by Andrea Piazza, Jaqueline Rafaela Travessini
System Logistics

Stacker cranes handle pallet placement in the warehouse, moving longitudinally along the aisles between the warehouse shelves to deposit and retrieve loads. They come in different heights and move along two towing rails, one on the ground and one overhead that is connected to the rack. Stacker cranes are susceptible to earthquakes that can damage them and the racks around them. As an important piece of equipment in the warehouse, any damage they suffer can impact heavily on restoration costs and can interrupt pallet flows causing further economic damage to the warehouse owners.
Each region of the world has different seismic norms that describe how to calculate the seismic load that structures may suffer in that location. These norms are intended to ensure the safety of structures, establish seismic intensity levels, and set minimum requirements for the seismic design of structures. The seismic load specification is based on determining the expected magnitude and intensity of earthquakes and their effect on the terrain at a given site, and converting these in some way into the parameters of ground motion. The stacker crane in this article will be installed in Mexico in seismic zone B.

Seismic load spectrum

A geotechnical study of the terrain in Mexico had been undertaken for the purpose of issuing the necessary recommendations for the design of foundations. The following values were extrapolated from this document:

$a_0 = 0.13*981 = 127.53 cm/s^2$

$c = 0.326*981 = 319.8 cm/s^2$

Where a_0 is the maximum ground acceleration and c is the design coefficient. Furthermore, according to the geotechnical study, the structure should be classified as Group B, Type 6.

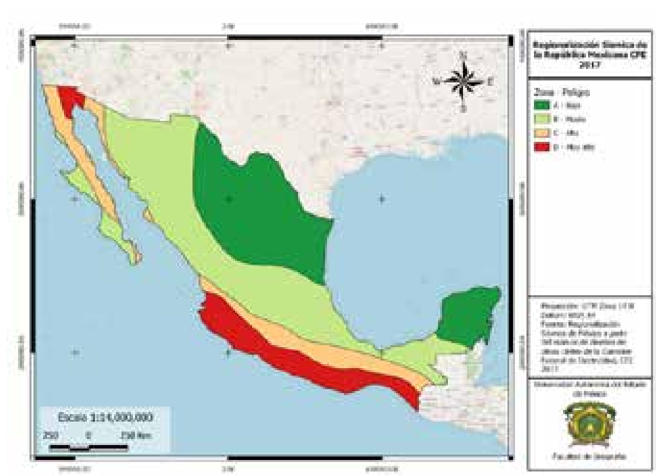


Fig. 1. Seismic regions of the Mexican Republic (source: CFE, 2017).

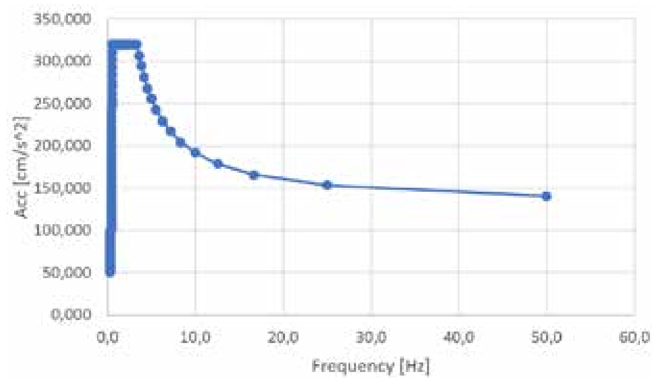


Fig. 2. Load Spectrum of terrain - classified as Group B – Class 1 – Zone B – Type III.

According to the manual the terrain on which the stacker crane operates is classified as Group B – Class 1 – Zone B – Type III, which has the following load spectrum:

Set up of the seismic analysis

The seismic analysis begins with a static structural analysis of the full structure and ends with a response spectrum analysis (RSA), following the workflow shown below:

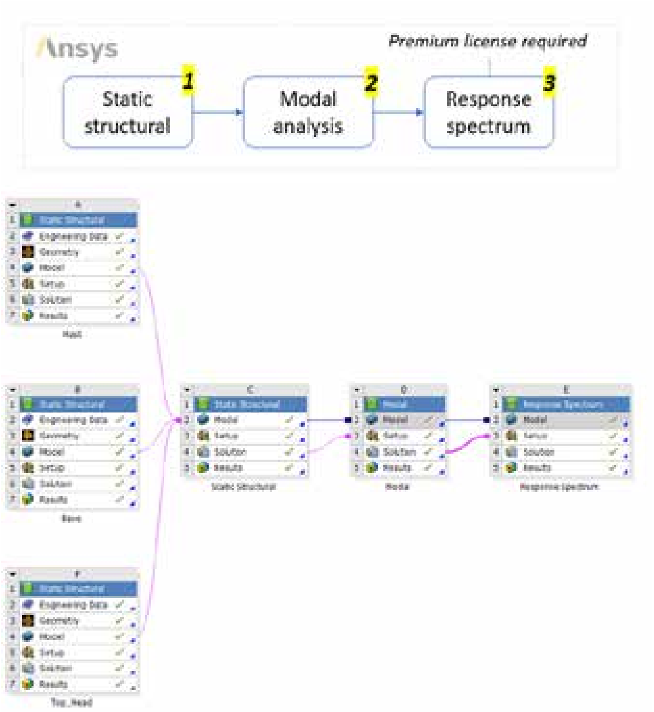


Fig. 3. Ansys Workbench workflow.

Static structure

Stacker cranes can reach up to 40m in height and 30 tons in weight. The first step in the workflow is to perform an analysis of the static structure with all its mass points, forces, constraints, at standard earth gravity. The static analysis provides the pre-stress levels and constraints that will influence the modal analysis. The complete 3D model of the piece of equipment is developed, including various details concerning the mast, base, and upper parts.

Modal analysis

Modal analysis is a dynamic analysis that generates the natural frequencies at which a structure will resonate. It allows engineers to avoid resonant vibrations or vibrations at a specific frequency and provides information on how the design is likely to respond to different types of dynamic loads. Modal analysis can be performed with or without pre-stress.

Pre-stress modal analysis is a study of the effect of loading on the dynamic response of a structure. Pre-stressing the structure can result in a significant variation in the frequencies of the extracted modal shape, especially at the first mode compared to higher modes.

For the purpose of this study, 45 modal shapes were extracted for the stacker crane, resulting in an effective mass ratio of more than 80% in the X, Y, and Z directions. The modal analysis was performed with pre-stress.

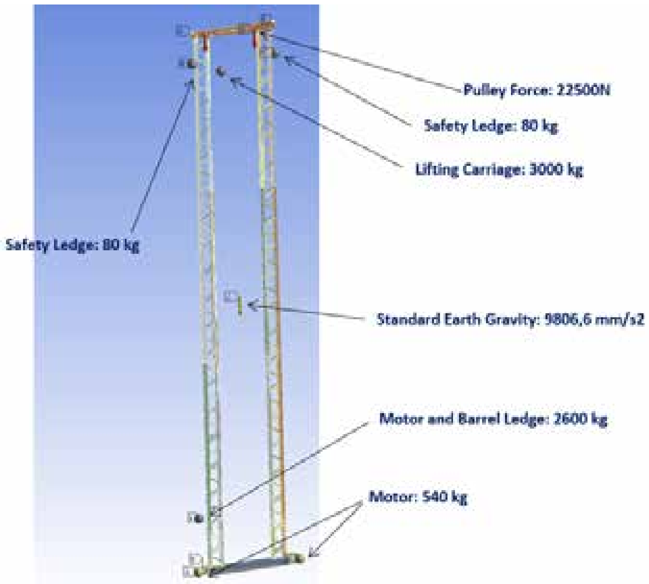


Fig. 4. Analysis of the static structure - mass point.

***** MODAL MASSES, KINETIC ENERGIES, AND TRANSLATIONAL EFFECTIVE MASSES SUMMARY *****

				EFFECTIVE MASS					
MODE	FREQUENCY	MODAL MASS	KENE	X-DIR	RATIO%	Y-DIR	RATIO%	Z-DIR	RATIO%
1	0.5914	6.493	43.32	10.42	62.41	0.1073E-03	0.00	0.4299E-08	0.00
2	1.015	4.947	100.7	0.8980E-05	0.00	0.2769E-03	0.00	11.62	87.54
3	1.157	4.154	115.6	0.2791E-04	0.00	0.2108E-03	0.00	0.1220	0.60
4	2.159	6.018	380.2	0.4780E-01	0.24	0.4104E-03	0.00	0.4580	2.16
5	2.672	4.532	346.6	0.9413E-01	0.47	0.3652E-03	0.00	0.1026E-03	0.00
6	2.974	4.163	479.7	0.9097E-01	0.45	0.4475E-03	0.00	0.7325	3.63
7	3.075	6.937	1294.	2.177	10.78	0.6711E-02	0.03	0.4461E-01	0.22
8	3.127	6.617	1123.	0.7901	3.91	0.1098E-02	0.01	0.3234	1.60
9	4.414	1.979	761.3	0.1446E-01	0.08	0.2576E-03	0.00	1.982	9.87
10	4.967	1.126	947.5	0.3981E-02	0.02	0.2142E-04	0.00	0.9585E-01	0.48
11	5.392	2.599	1482.	0.9142E-01	0.45	0.1819E-03	0.00	0.9314E-01	0.46
12	5.573	3.644	2354.	0.8778	4.35	0.3387E-03	0.00	0.7097E-02	0.03
13	5.879	1.270	866.1	0.1700E-03	0.00	0.8970E-04	0.00	0.6962E-03	0.00
14	6.605	3.248	2795.	0.3539	1.75	0.2229E-01	0.11	0.6856E-03	0.00
15	6.816	3.664	3360.	0.5439	2.69	0.1318E-01	0.07	0.7627E-02	0.04
sum				16.39	81.16	16.49	81.66	18.61	92.16

Fig. 5. Modal analysis - effective mass.

Response spectrum analysis

A response spectrum analysis calculates the maximum response of a structure to a transient load. It is performed as a rapid alternative to a full transient solution. The load can consist of a displacement, velocity or acceleration. It is possible to apply the same load to all constraints (single point case), or different loads to each constraint (a multi-point case). The response spectrum analysis technique combines the response spectrum of a given dynamic load with the results of the modal analysis. These maximum modal response values are then combined using a method (such as sum of absolute values or complete quadratic combination, CQC) to estimate the peak response of the structure.

For periodic modal responses with sufficiently separate frequencies, the square root of the sum of the squares (SRSS) method is the correct method to combine the modal responses. Where the modes have closely spaced frequencies, various conservative methods such as complete quadratic combination (CQC) and Rosenblueth (ROSE) can be used to combine the modal responses - the SRSS method is not suitable in this case.

The definition of modes with closely spaced frequencies is a function of the critical damping ratio. At critical damping ratios ≤2%, modes are considered closely spaced if their frequencies are within 10% of each other (i.e. for $f_i < f_j$, $f_j \leq 1.1\, f_i$). At critical damping ratios >2%, modes are considered closely spaced if their frequencies are within five times of each other's critical damping ratio (i.e. for $f_i < f_j$ and 5% damping, $f_j \leq 1.25\, f_i$; for $f_i < f_j$ and 10% damping, $f_j \leq 1.5\, f_i$).

Mode	Frequency	fj ≤ 1.1 fi	
1	0,5814		
2	1,015	0,63954	Ok
3	1,187	1,1165	Ok
4	2,189	1,3057	Ok
5	2,472	2,4079	Ok
6	2,874	2,7192	Ok
7	3,075	3,1614	closely spaced
8	3,127	3,3825	closely spaced
9	4,414	3,4397	Ok
10	4,967	4,8554	Ok
11	5,392	5,4637	closely spaced
12	5,573	5,9312	closely spaced
13	5,879	6,1303	closely spaced
14	6,605	6,4669	Ok
15	6,816	7,2655	closely spaced

Table 1. Closely-spaced frequency verification.

Fig. 6. Deformation RSA.

Since there are many closely spaced frequencies, the SRSS method is not applicable in this case. The ROSE method, which offers a mathematical approach to modal correlation evaluation for seismic response spectrum analysis, was chosen to evaluate the displacement, stress, and reaction forces of the model.

RSA reaction forces

The RSA results allow the reaction forces to be calculated and considered by the designers of the complete rack structure. According to the RSA analysis, the reaction forces on the upper wheels of the stacker crane are: A: 7733,7 N; B: 10478 N

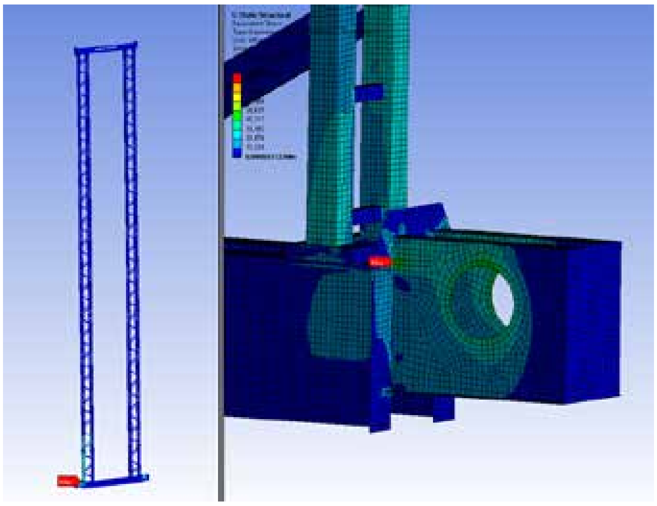


Fig. 7. Equivalent stress and maximum stress.

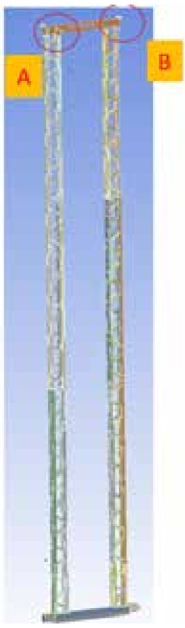


Fig. 8. Reaction force – upper wheels.

Conclusion

Simulation procedure, as shown, allows a safe and reliable design of stacker crane fully integrated in a whole warehouse system. Risk related to seismic occurrences is drastically reduced and evaluated. it was possible to analyse the results in terms of the stress and reaction forces and to verify that they are compliant with System Logistics standards.

At the given maximum acceleration of 0.13g, the stacker crane's maximum stress does not reach critical values, having a safety factor of 1.53. Consider the crane is 38.5 meters high, the deformation of 54mm only accounts for 0.14% of the total height, which is also acceptable.

The response analysis enables both the stacker crane and the rack to be correctly dimensioned, guaranteeing optimal safety for the warehouse and helping the manufacturers of both the stacker crane and the rack to deliver a better product to the customer.

About System Logistics

System Logistics is a leading global supplier of innovative intralogistics and material handling solutions to optimize operations in warehouses, distribution centres, and manufacturing supply chains worldwide. With a special focus on the Food & Beverage and Grocery sectors, System Logistics develops tailor-made solutions for automated storage and picking, including stacker cranes, material handling equipment, software, and services. Most of the technologies we use in our solutions are fully engineered, developed, and manufactured in-house and are then also installed by System Logistics. This gives us complete control of the process. We are a dependable, long-term partner: collaborating closely with our customers, we develop flexible solutions that address the challenges they face every day and that will grow and change with them in the future.

For more information:

Andrea Piazza – System Logistics
andrea.piazza@systemlogistics.com

References

[1] Seismic regions of the Mexican Republic (source: CFE, 2017).

Workshop at HiPEAC 2024

Plasma Physics
Towards the ExaScale Era

January 19th, 2024 | 10.00 – 16.00
Science Congress Centre
Munich, Germany

Plasma constitutes the fourth state of matter and is by far the most mysterious with its inherently nonlinear, multidimensional, and multiscale dynamics, making theoretical modelling somewhat limited. Some great progress in plasma physics has been achieved using computer simulations, while advances in High Performance Computing (HPC) have enabled us to further boost our knowledge in this field.

The advent of the exascale era, however, poses some important challenges that are impeding this positive trend – largely because of the requirement to adapt existing plasma codes to these novel exa architectures.

Recognizing these challenges, the European High Performance Computing Joint Undertaking this year funded two Centres of Excellence (CoEs) to prepare codes for dealing with different aspects of plasma physics for the exascale era and beyond.

SPACE

PLASMA PEPSC

This HiPEAC 2024 workshop will bring together plasma researchers, code developers, research software engineers, and HPC experts from the newly launched SPACE and Plasma-PEPSC CoEs, as well as the wider plasma code community to discuss key issues and possible solutions for porting selected codes to new emerging architectures.

www.space-coe.eu | info@space-coe.eu

Co-funded by the European Union

EuroHPC Joint Undertaking

Centres of Excellence for HPC Applications – Horizon-EuroHPC-JU-2021-COE-01

Funded by the European Union. This work has received funding from the European High Performance Computing Joint Undertaking (JU) and Belgium, Czech Republic, France, Germany, Greece, Italy, Norway, and Spain under grant agreement No 101093441

Simulation and integration of a DED repair pipeline into the machine tool control environment

by Daniele Calsolaro, Livio Airaudi
EnginSoft

The terms "3D printing" or "Additive Manufacturing" (AM) refer to the creation of solid three-dimensional objects from a digital file by depositing successive layers of a material until the object is created. Each deposited layer can be viewed as a narrow cross-section of the final object. There are several types of material deposition process, such as powder bed fusion, binder jetting, material extrusion, material jetting, vat polymerization, and direct energy deposition.

Directed energy deposition (DED), more commonly called metal DED, is an AM technology related to metals, in which a metal in powder or wire form is melted using a high-density energy source and is simultaneously deposited locally to form the object in question.

DED technology can be used to repair existing components or to manufacture large new components. While it can be used to manufacture parts from scratch, the technology is mostly used in industry to repair large costly components like turbine blades or propellers that have been damaged during use. As a result of the benefits accruing from its use, including cost and scrap reduction, shorter repair times, and minimization of inventories, and the related substantial environmental benefits, this technology is rapidly gaining in importance.

The EU EIT Manufacturing's OScaR project, which lasted from January to December 2022, was a collaborative undertaking to define a technological solution for repairing metal components using DED AM. Industrial manufacturers require an automated or semi-automated solution that enables them to mount a part that has a partially unknown shape on a machine, measure the relevant part of its surface in 3D, and automatically generate an AM repair program to return the part to the desired shape.

The OScaR project

A typical DED machine consists of a multi-axis instrument with a nozzle that deposits molten material onto the work surface – a base or component – that has been firmly clamped. The metallic material is fed to the nozzle as a powder or wire. This material is melted during deposition by a concentrated heat source - usually a laser electron beam or plasma arc.

In principle all weldable metals such as titanium and its alloys, Inconel, tungsten, stainless steel, and aluminium can be 3D printed with DED. The size of the wire used typically ranges from 1-3mm in diameter, while the powder particle sizes are similar to those used in powder metallurgy processes, i.e. between 50 and 150 microns.

Obviously, such a process generates temperatures with very high thermal gradients that can lead to significant overheating, deformation, and accumulation of residual stresses during the layer deposition phase; the deformation effects can be so great that they can affect the proper functioning of the technology and cause fracture or breaking.

Therefore, it is extremely important to be able to accurately predict, specifically by using finite element simulation, the effects of DED printing related to deposition, with the aim of optimizing the printing process in terms of time and to reduce the residual stress.

End users need an automated or semi-automated solution to mount a part with a partially unknown shape onto a machine, measure the relevant part of its surface in 3D, and automatically generate a repair program to restore it to the desired shape. The generated part must be automatically analysed to ensure inline quality control. Furthermore, since the repair process does not allow for failures, the use of a simulation tool is mandatory to pre-assess the results that can be obtained.

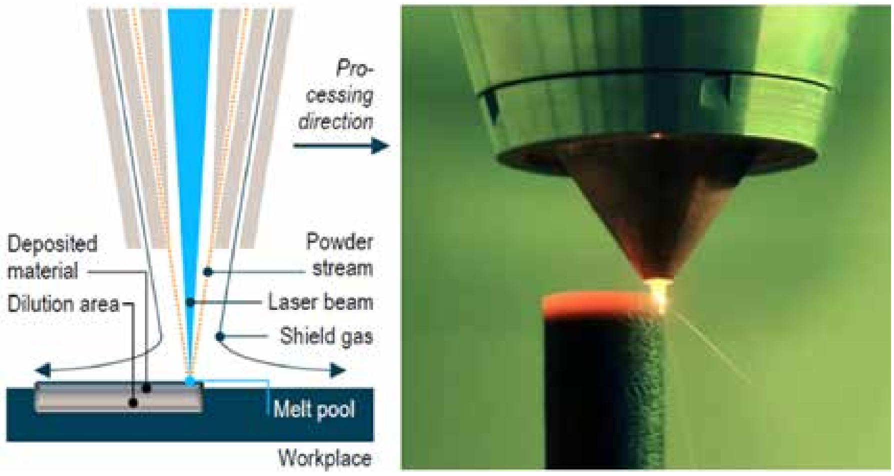


Fig. 1. Direct energy deposition (DED).

In many modern manufacturing environments, in-situ reconstruction of a damaged or faulty part's geometry is a highly desired function in the production pipeline, for example, damaged parts can be repaired/reworked to extend their life cycle. However, working on a damaged part requires applying the preferred remanufacturing or repair technology to a surface with a partly unknown shape.

Parts, especially those prefabricated externally, that have to move through multiple machines during production, also have this requirement for repair or remanufacture. Many industries would benefit from the ability to easily modify multi-process parts and repair damaged parts, but lack an established technology to 1) easily work on existing parts that have a far from nominal geometry, and 2) automatically generate an AM-DED repair toolpath for a damaged segment. Furthermore, quality control of the generated part must be performed entirely off-machine, requiring additional production time.

The OScaR (Optical SCAn-and-Repair solution for machine tools) project, co-funded by the EU's EIT Manufacturing, focused on the use of DED technology; in particular, the project's ultimate goal was to define a technological solution to repair metal components. OScaR enables the repair of complex metal parts for high quality applications using an AM-DED machine, in-situ 3D scanning, DED simulation and toolpath generation to achieve the next level of flexible manufacturing.

The project's most important impacts on the manufacture-and-repair industry worldwide are to enable:

- Environmentally sustainable production
- Lower energy consumption
- Reduced consumption of raw materials
- Reduced engineering time through automation
- Inline quality control of processed parts

Simulating the DED repair process

EnginSoft's role was to develop a suitable simulation method and configuration to virtually replicate the DED process for repair applications, smoothing the path for new potential customers. The model to simulate the DED repair process is based on Ansys Additive Suite and enables the effects of the repair process on the reference part to be predicted and optimized. The Ansys DED simulation module imports a scanned baseline geometry and an externally generated G-Code. The main materials and process configurations relevant to the use cases are entered into the simulation parameters. The final simulation of the deposition process and its thermo-mechanical deformation was developed and tested on two principal use cases, which included validation of the results against real measurements.

Ansys Additive Suite is a powerful collection of tools from Ansys specifically for additive manufacturing simulation. Workbench Additive is one of the tools designed for use within

Fig 2. A-frame test case.

the Workbench platform and Mechanical application. The DED Process Simulation functionality in Workbench Additive is implemented as an ACT extension that must be loaded into Workbench. The objective of DED Process Simulation in Workbench Additive is to predict temperature-induced deformations and stresses in the various components on a macroscopic level during the production phase to prevent failures, while simultaneously providing trend data to enable improvements during the design phase of the additive process, including the orientation of the parts and their build order.

To simulate the DED manufacturing process, the analysis must follow the actual printing process as it is deposited and solidifies, so-called "track-by-weld" solidification. In this type of simulation, the thermal analysis and structural physics (stress and distortion) tend to be decoupled which allows the full thermal process to be simulated prior to the structural simulation. In a DED process simulation, the model evolves over time with elements being added during the process.

To begin, the full initial part is meshed using Cartesian or tetrahedral elements and then a "birth and death" technique is applied, which allows the sequential activation of element clusters to simulate the progression of the print job (where the term "cluster" defines a part of the weld path). In addition, the associated boundary conditions for each stage also develop as thermal convection surfaces. The build phase is complete when all the elements have been activated (brought to life).

Analysis times and time steps are controlled by the process parameters and are not known in advance; these aspects are verified internally during the simulation phase. The DED process requires a very detailed level of simulation using real weld lines and an abstraction known as element grouping.

This grouping is used to sub-divide the weld lines into smaller portions of mesh, called clusters, to which the thermal conditions are sequentially applied at each time step. Each cluster is thus a portion of a weld line that is created sequentially by activating that section through the birth-and-death technique and assigning to each newly activated piece the thermal conditions resulting from the thermal simulation performed on the previous piece.

The project involved an initial configuration phase using an example case (A-frame) after which the optimized method was applied to repair a turbine blade.

Simulation setup is guided by a wizard to ensure that all the necessary steps are followed to discretize the model and for the subsequent simulation. In particular, the wizard guides the user through the following steps:

- **Import Geometry:** defining the bodies to be printed;
- **Mesh Creation** (a hexahedral mesh is recommended): the DED Process does not require a strictly layered mesh of identical layer heights, but the weld trace must be represented in the mesh. A slightly coarser mesh is acceptable for the base plate because it simply serves as a heat sink and fixed support in the simulation. Mesh types to consider include Cartesian, tetrahedral, and sweep meshes. A hexagonal (i.e. Cartesian) mesh should be preferred to a tetrahedral mesh if manual clustering is to be used because some tetrahedra may be excluded from element clusters for some geometries. We also recommend using linear rather than quadratic elements;
- **Clustering:** it is possible to create a cluster manually or to use a G-Code file, as in our case. The cluster volume (in mm^3) is used to control the cluster size and therefore has a direct influence on the simulation time. This value determines how many elements are activated per loading step; the time for this loading step is then determined by volume/deposition rate. A smaller cluster volume tends to give a more accurate result. Depending on the overall size of the build geometry, this value should be determined by balancing the computing effort with the desired accuracy. Set the cluster volume according to the total volume of your part;
- **Material assignment:** in this window, you can assign the material to be printed and the basic components;
- **Define build settings:** in this step the user defines the machine and process settings and conditions, grouped according to three categories:
  - Machine Settings: The process parameters, which vary for each DED machine and according to the material used in the deposition process;
  - Material Deposition Rate: The feed rate of the molten material, in mm^3/sec. This value can be determined by multiplying layer thickness (mm) x weld width (mm) x deposition speed (mm/sec); and
  - Build Conditions: The settings for the environment in the build chamber that surrounds the part during printing, including the preheating temperature;
- **Boundary conditions:** in this last step the user defines the constraints of the

model in detail and, more specifically, the bottom of the base is fixed to the ground;

- **Generate clusters:** before generating the clusters, the position of the G-code can be checked using the "Show path" option in the multifunction bar (green line for laser on, blue line for laser off).

The two DED case studies

Following the setup described above, the transient thermal simulation was performed for the two case studies. Typical significant results are temperatures in the transient thermal solution, and displacements and equivalent stresses in the static structural solution.

The results obtained in terms of temperatures and equivalent Von Mises stress for the "A Frame" case study are shown in Fig. 3; the temperature and residual strain values were compared with the actual results obtained by one of the partners, SUPSI, using Prima Additive's DED machine, thus validating the Ansys results.

The full procedure was then repeated on an Inconel718 blade (the second case study) in order to verify that the method would work for an industrial case. In fact, this second structure is a perfect example of a major potential application for this method.

Conclusions

The solution for repairing complex metal parts was developed, tested, and validated as part of the OScaR project. This complete part-repair method combines an AM-DED (directed energy deposition) machine, in-situ 3D scanning, DED simulation and toolpath generation to achieve the next level of flexible manufacturing.

The simulation tool pre-assesses the results that can be obtained by using additive manufacturing processes in the repair pipeline to get the part right the first time.

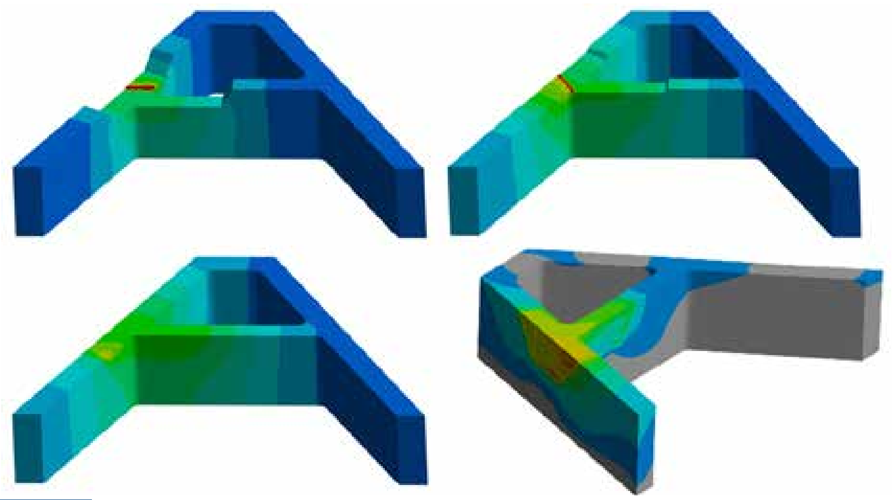


Fig 2. A-frame test case.

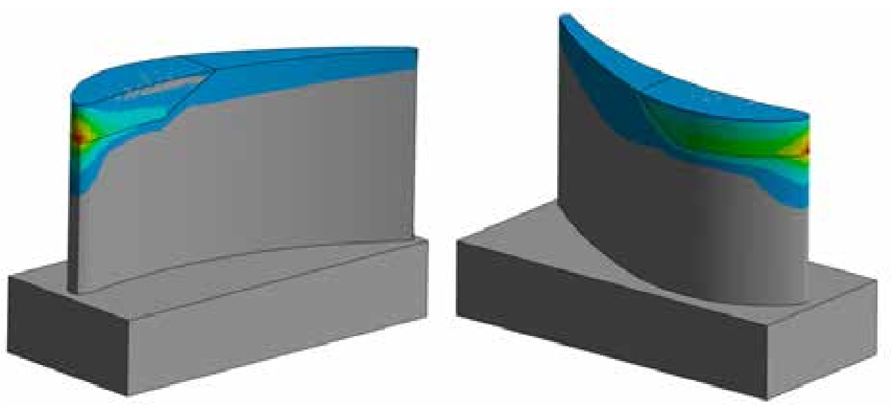


Fig. 4. Simulation and results of the DED repair process applied to the tip of a turbine blade.

In summary, OScaR automates metal part repair through an advanced inspection and toolpath generation platform. The advantages for manufacturers and users are:

- An enormous reduction in the engineering time required to develop such repair strategies;
- The ability to repair a wider range of complex parts;
- Increased ease of repair compared to manufacturing parts from scratch; and
- Consequently, cost reduction throughout the production chain.

The results achieved are important for society because of:

- Environmentally friendly production (several large parts were repaired instead of replaced, as is currently common practice);
- Reduced energy consumption;
- In-line quality control of the processed parts.

In addition, the OScaR project

- Supported the development of a dedicated ecosystem for new services and value chains;
- Created a robust solution to enable the provision of high-quality manufacturing services for research and consultancy
- Enabled increased competitiveness.

Acknowledgements

The OScaR project was co-funded by the 2022 EIT Manufacturing Call for Proposals for "Flexible production systems for competitive manufacturing". We would like to thank all project partners (Prima Additive, SUPSI, inspire, AIDIMME, POZZO, and EnginSoft) for their cooperation and contribution to the success of the project.

For more information:
Daniele Calsolaro – EnginSoft
d.calsolaro@enginsoft.com

Article from: *Futurities* Special Issue 2023 - Additive Manufacturing

LIFESAVER
Living Impact on Fetal Evolution

Human placenta reproduced in a laboratory

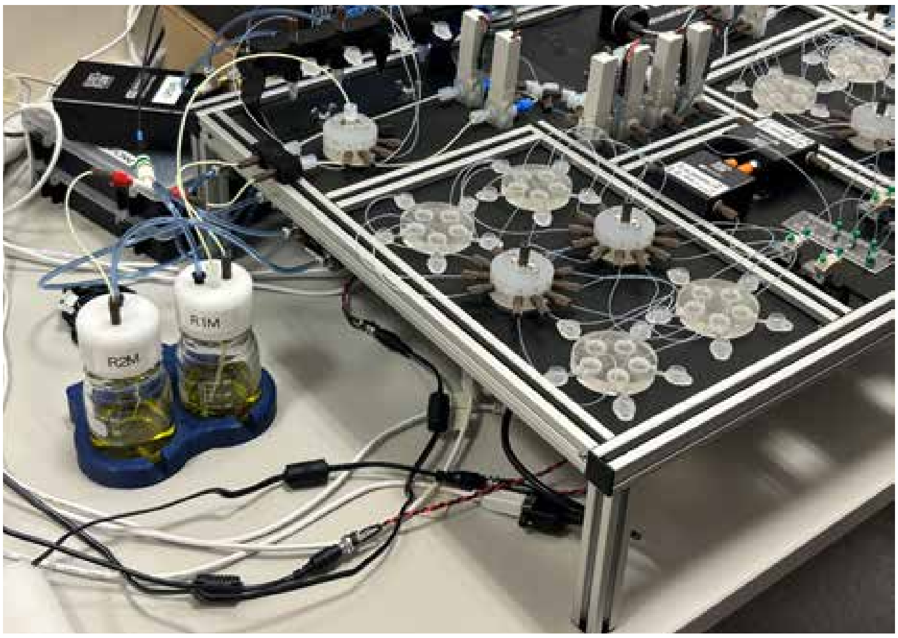
The LIFESAVER project prototype has been birthed

by Carla Baldasso
EnginSoft

Pollution by microplastics, chemicals and antibiotics is one of the major environmental problems of the last ten years and inevitably affects human health, including of the unborn.

To date, little is known about what happens to a pregnant woman or her foetus when they are exposed to such pollutants or when the mother-to-be takes a medicine, specifically how much of these chemicals reach the foetus.

The fundamental role of the placenta is to protect the foetus and to allow vital substances, such as nutrients and oxygen, to pass from mother to child. Unfortunately, it is well-known that, along with substances vital to the foetus, other potentially harmful substances are also passed on. The proof of this is the fact that microplastics have been found in the placenta.



Since testing on pregnant women is justifiably prohibited to protect the health of the mothers and of their unborn children, and testing on animals is meaningless because their placentas have a different structure and behave differently, there is no existing knowledge or technology that can predict the transmission of chemicals from the mother to her baby in the womb.

The EU's LIFESAVER project, coordinated by EnginSoft, is developing a laboratory system to forecast the penetration of various chemicals through the placenta. The first prototype was presented in Braga in Portugal at the International Iberian Nanotechnology Laboratory in June 2023.
This system, unique in the world, replicates both the mother's blood circulation and

the foetus's blood circulation through two microfluidic circuits.

A membrane, housed on a chip and representing the placenta, lies between the two circuits. The high-tech system is equipped with various nano-biosensors capable of detecting concentrations of harmful substances such as antibiotics, antivirals, hormones, or microplastics in the two circuits as well as their transfer from one circuit to the other.

Top European experts from the best in class European universities, research institutes and companies have been involved in creating this prototype. It was the transdisciplinary nature of the expertise – in microfluidic systems, 3D organ printing, cell cultures, nano-biosensors, and digital twins – that made it possible to realize such a complex system.

The main challenge of the project is to ensure the system biofidelity, i.e. to replicate, as closer as possible, the placenta's behaviour during the first three months of pregnancy, which are the riskiest for foetal development.

The placenta is the most complex organ in the human body and the most difficult to study due to the enormous transformation it undergoes day by day during pregnancy. While some literature exists on the placenta at term, nothing or little is known about the placenta during the first three months, both due to the ethical issues concerning voluntary abortion and to the fact that the placenta is completely destroyed and unusable for scientific research in the event of a miscarriage.

Therefore, the only way to create a model of the placenta's behaviour is to reproduce it virtually, through a bio digital twin.

The digital model, or digital twin, consists of a series of fluid-dynamic models that emulate the blood flows in and across the placenta; machine learning algorithms make it possible to correlate these flows with the health of both the maternal and foetal cells.

However there is still a long way to go from this initial prototype to achieving a stable, reliable and easily reproducible system: the researchers have two more years of work ahead of them to fine-tune the system, validate its functionality, and create the protocols for its use.

At present, every package insert for any medication states "Do not take during pregnancy", or "Take under strict medical supervision during pregnancy".

The Lifesaver projects wants to overcome this and will culminate in a system to certify which medicines are safe and which are not during pregnancy. The LIFESAVER vision is for every pregnant woman to have a proper living environment with minimal risks to the fetus, and to be safeguarded with scientifically justified regulations of potentially harmful chemical and medicinal products. This will lead to a healthier quality of life for the mothers and for their babies.

This project has received funding from the European Union's Horizon 2020 Research and Innovation Programme under Grant Agreement No. 101036702. The project supports the implementation of the European Green Deal with regard to the study of exposure to industrial chemicals and pharmaceuticals.

For more information:

Carla Baldasso - EnginSoft
c.baldasso@enginsoft.com
or visit lifesaverproject.eu

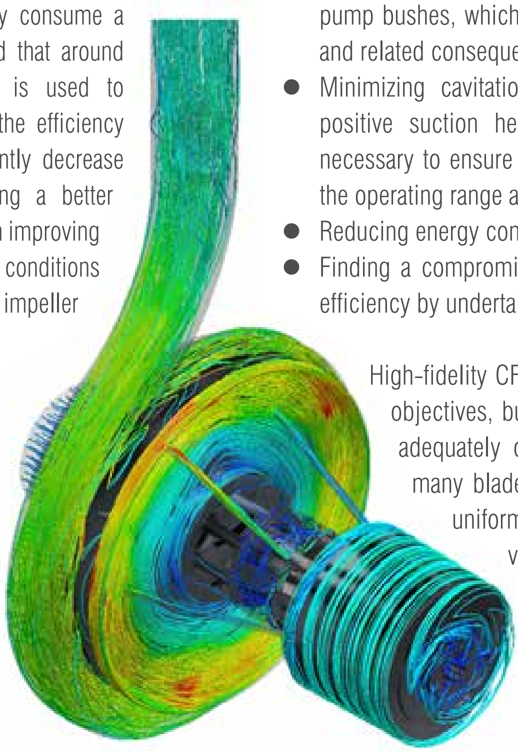
Cloud-based CFD Optimization of Magnetic Drive Pumps using HPC

by Alessandro Arcidiacono[1], Marisa Zanotti[1], Luigi Fossati[2]
1. EnginSoft - 2. CDR Pompe

Mechanical pumps play a significant role today due to their widespread use in sectors as diverse as energy, oil&gas, aerospace, automotive, heating, ventilation, air conditioning (HVAC), and consumer goods. From an industrial perspective, the most common type are centrifugal pumps. Magnetically coupled pumps, or magnetic drive pumps, are a particular subset of centrifugal pumps and differ from conventional pumps because the motor is connected to the pump via magnetic couplings instead of a direct mechanical shaft. The chemical, pharmaceutical and nuclear industries make use of aggressive or hazardous fluids, and therefore need magnetic drive pumps, which eliminate the need for shaft seals, to comply with safety regulations and prevent the escape of these fluids.

Since centrifugal pumps are widely used, they consume a considerable amount of energy: it is estimated that around 10% of the electricity produced worldwide is used to operate pumps. Even small improvements to the efficiency of centrifugal pumps would therefore significantly decrease overall energy consumption. However, creating a better design for a centrifugal pump does not only mean improving its efficiency: it must avoid creating cavitation conditions that could cause noise, vibration, and ultimately, impeller failure.

Magnetically coupled pumps are extremely sensitive to axial thrust due to wear-and-tear problems during their life cycle. Without the support of a mechanical shaft to stabilize the structure, it is imperative to balance the impeller, the magnetic drive system, and the secondary flows of axial forces over the entire operating range of the machine. The consequences of a poor design can lead to machine failure.



The challenge

Magnetic-drive chemical-process pumps eliminate the need for shaft sealing thus reducing costs and improving safety. These pumps are used to prevent the leakage of the aggressive fluids used in the chemical, pharmaceutical or nuclear industries.

CDR Pompe wanted to re-design all four sizes of this type of pump in order to improve their performance, reduce the design and production costs, and deliver more efficient and more competitive products to the market. The following improvements were considered to be the most important:

- Minimizing axial forces (to avoid too much thrust against the pump bushes, which could lead to premature machine failure and related consequences);
- Minimizing cavitation, thereby decreasing the NPSHr (net positive suction head required) i.e. the suction pressure necessary to ensure the pump operates correctly, to increase the operating range and flexibility of use;
- Reducing energy consumption;
- Finding a compromise between these objectives and greater efficiency by undertaking a multi-objective analysis.

High-fidelity CFD makes it possible to envision all these objectives, but a very detailed approach is required to adequately characterize this type of machine where many blades feed the downstream volute in a non-uniform manner and secondary flows influence volumetric efficiency, NPSHr and axial thrust. This must be done without simplifying the assumptions, therefore explicitly modelling all losses and sections of the magnetic drive. However, these simulations are CPU-intensive due to the complex geometries and multi- meshes required to model the machine in the actual operating scenario.

The solution

In the experiment, CDR worked with EnginSoft and used CFD modelling and optimization for the four pump sizes to find the best compromise between geometric fidelity, accuracy, and computational costs.
Mesh sensitivity analysis and pressure evaluation of the CFD models led to a high-fidelity steady-state approach with a suitable mesh (30-70M elements). The subsequent geometry-based optimization process, built on top of the CFD model, was based on a design of experiment (DoE) and response surface methodology. The purpose of a DoE is to collect a representative set of data to calculate a response surface and then perform an optimization. Basically, a set of design points is collected or calculated. The accuracy of the response surface is highly dependent on the DoE scheme, and on the number of design points calculated.

According to CDR's redesign requirements, a total of seven input parameters had to be considered. An accurate DoE scheme with this number of parameters required the calculation of approximately 100 design points. Considering four pump sizes, the multi-scale and multi-scenario optimization process required around 400 large model simulations to be performed.

This methodology resulted in optimized performance maps for each pump size in terms of head, efficiency, power, NPSHr (down 20-25%, reducing the risk of cavitation) and axial thrust (down 5-20%, allowing longer pump lifetime). The results led to an increased ability to operate the pumps in extreme situations that were expected to be critical, such as conditions at the limit of cavitation or similar, and provide a good balance between efficiency and thrust.
For all these reasons, an overall study and optimization of this machine (multiscale and multi-scenario) required access to high-performance computing (HPC) infrastructure and the use of specific engineering simulation tools. EnginSoft provided the engineering knowledge in terms of CFD modelling, while CINECA provided the hardware and support needed to conduct the CPU-intensive parallel calculations.

Business benefits

In addition to making CDR more competitive in the European market, the improved models, being ASME compliant, will help it access the US market, with a consequent growth in the company's expected annual turnover of 1.5M€ in five years. The reduced maintenance requirement is a game-changer for an SME that lacks maintenance support staff in the US and is therefore a competitive resource in such a complex market. In addition, the HPC-based design process reduces time-to-market by 50%.

The reduced energy consumption also translates into lower CO_2 emissions. An improved magnetic drive pump design prevents fluid evaporation and reduces noise and vibration, resulting in greater versatility, reliability, and pump durability, with less planned and unplanned downtime and less maintenance required.

Experiment partners

CDR Pompe is a leading Italian manufacturing SME that has been developing and producing

pumps for hazardous, corrosive, and highly pure liquids in the chemical, pharmaceutical and other process industries since 1978.

EnginSoft is a consulting SME specializing in the field of computer-aided-engineering (CAE) with offices around the world. Founded in 1984, EnginSoft today has more than 160 employees, 10 offices, and partnerships with companies, R&D centres, and universities.

CINECA - Consorzio Interuniversitario, founded in 1969, is Italy's largest supercomputing centre with an HPC infrastructure equipped with cutting-edge technology and highly qualified staff working with academic and industrial partners.

The FF4EuroHPC project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No. 951745. The JU receives support from the European Union's Horizon 2020 research and innovation programme and from Germany, Italy, Slovenia, France, and Spain.

For more information:
Alessandro Arcidiacono - EnginSoft
a.arcidiacono@enginsoft.com

The success story presented in this article was developed during the first tranche of FF4EuroHPC Project. FF4EuroHPC supports the competitiveness of European SMEs by funding business-oriented experiments and promoting the uptake of advanced HPC technologies and services. The experiment is an end-user-relevant case study demonstrating the use of cloud-based HPC (high-performance computing) and its benefits to the value chain (from end-user to HPC-infrastructure provider) for addressing SME business challenges that require the use of HPC and complementary technologies such as HPDA (high performance data analytics) and AI (artificial intelligence). The successful conclusion of the experiment created a success story that can inspire the industrial community.

High-fidelity modelling of a small wind turbine

by Sotiris Kokkinos[1], Dimitrios Lampropoulos[1], Vasilis Papatsiros[2]
1. FEAC Engineering - 2. Eunice Wind

Wind energy is one of the fastest-growing renewable energy sources in the world generating a global capacity of more than 837GW in 2022. As the demand for clean energy continues to grow, the wind industry faces greater pressure to improve the efficiency and cost-effectiveness of wind turbines. One way to do this is by reducing the costs associated with research and development which often involve physical testing.

The challenge: running a complex wind turbine simulation

The use of high-fidelity numerical models facilitates the accurate simulation of the aerodynamic performance of wind turbines in a range of operating scenarios. However, to accurately predict temporal and spatial scales within the multi-scale CFD (computational fluid dynamics) model requires advanced meshing techniques and fine discretization.

The objective of this experiment was to build an accurate numerical model of Eunice's EW16 wind turbine (Fig. 1) to improve its operational performance. These simulations however require significant computing power and expertise that exceed the company's inhouse resources. FEAC Engineering was appointed to assist Eunice to address the challenge using its knowledge and expertise in the simulation sector. For FEAC Engineering, success would allow it to capitalize on emerging opportunities in the HPC (high performance computing) simulation market, thereby expanding its reach and impact in the field.

The solution: digital twin – a deeper understanding of the turbine's performance

The sophisticated CFD simulation models developed by the partners harnessed the power of HPC resources to test and optimize the performance of the wind turbine at its operation site. On average, more than 485,000 CPU hours were used for the simulations. By using the available HPC infrastructure and adhering to the turbine manufacturer's requirements the partners were able to generate detailed data on the instrument's performance under varying wind speeds.

Satellite data was used to model the terrain and elevation changes at the location to enrich the information on the turbine's operational environment and was included in the simulation process. The simulation of the wind turbine's aerodynamics included computational meshes of over 50 million cells, a reduced time step approach, and the application of the detached eddy simulation method to model complex physical phenomena (Fig. 2). An experimental data set was used to validate the simulation procedure significantly reducing the uncertainty associated with the calculated results.

Improved emerging designs and more advanced wind turbine models

The simulations saved Eunice Wind many hours of physical testing and measurement. It costs

Fig. 1. The EW16 wind turbine.

the company about €300,000 in materials and licensing to build a new wind turbine exclusively for testing purposes. By replacing these tests with HPC-based CFD simulations that analyse the turbine's performance in its actual position, these costs were reduced by 10-17%.

The simulations uncovered tools and strategies that can further reduce the expense of R&D efforts in future development of the EW16. A potential 83% cost reduction is possible compared to an actual test installation due to the ability to identify the optimal settings and evaluate the stresses on the blades that are indicative of potential fatigue-related malfunctions. The simulation results can also help reduce expenditure through predictive maintenance, supporting more rapid design, and the manufacture of new blades using improved processes (such as novel resin materials or innovative airfoils for the EW16’s blades, etc.) to give Eunice a competitive advantage.

Through this project, FEAC Engineering gained valuable experience in high-fidelity simulations using large-scale HPC resources, which strengthened its expertise in numerical simulation and advanced computing and enabled it to expand into new markets, such as the wind energy sector.

The application of the new HPC-based methodology will also have a positive social impact, improving emerging design and advancing wind turbine models to contribute to environmental sustainability and avoid the unnecessary waste of the non-recyclable composites used in the construction of the test turbines. A final important aspect of the project is the European Commission’s recognition of wind energy as a top strategic sector and the fact that it is part of the “European Green Deal”, which strengthens the potential benefits of this project.

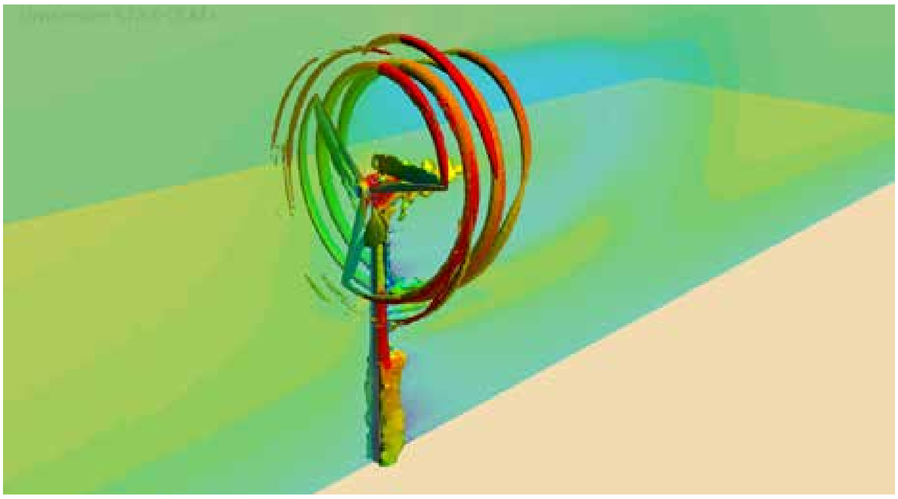


Fig. 2. Structure of the generated vortices.

Business benefits: 83% reduction in testing costs

HPC simulations have shown remarkable potential for cost reduction, offering substantial benefits over traditional methods. By replacing physical tests with HPC-based simulations, the expenditures associated with typical installations for testing purposes were reduced by an impressive 83%. The outlays on HPC simulations themselves vary depending on the complexity of the study being conducted and range from 10—17%, making it a flexible and cost-effective solution. In addition, by taking advantage of the HPC’s computing power, the integration of HPC simulations offers other significant advantages such as accelerating design cycles, and faster more efficient development of turbines. This reduction in operating costs then leads to more stable electricity costs. Furthermore, the adoption of HPC-based simulations aligns with the European Commission’s guidelines for a green and sustainable future. With the energy industry increasingly focusing on environmental sustainability, the partners’ use of advanced technologies such as HPC demonstrates their commitment to promoting renewable energy sources and reducing carbon emissions.

This success story was developed during the first tranche of FF4EuroHPC success stories. FF4EuroHPC supports the competitiveness of European SMEs by funding business-oriented experiments and promoting the adoption of advanced HPC technologies and services. The experiment is an end-user-relevant case study demonstrating the use of cloud-based HPC and the benefits it provides in the value chain from the end-user to the HPC-infrastructure provider, therefore it must address SME business challenges using HPC and complementary technologies such as HPDA (high performance data analytics) and AI. When the experiment is successfully completed, it results in a success story that inspires the industrial community.

The FF4EuroHPC project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No. 951745. The JU receives support from the European Union’s Horizon 2020 research and innovation programme and from Germany, Italy, Slovenia, France, and Spain.

For more information:
Tina Crnigoj Marc - Arctur
FF4EuroHPC Communication lead
ff4eurohpc@hlrs.de

The success story presented in this article was developed during the first tranche of FF4EuroHPC Project. FF4EuroHPC supports the competitiveness of European SMEs by funding business-oriented experiments and promoting the uptake of advanced HPC technologies and services. The experiment is an end-user-relevant case study demonstrating the use of cloud-based HPC (high-performance computing) and its benefits to the value chain (from end-user to HPC-infrastructure provider) for addressing SME business challenges that require the use of HPC and complementary technologies such as HPDA (high performance data analytics) and AI (artificial intelligence). The successful conclusion of the experiment created a success story that can inspire the industrial community.

Particleworks 8: the most advanced mesh-less fluid dynamics simulation software

by Michele Merelli
EnginSoft

The latest version of Particleworks, the particle-based fluid dynamics simulation software that uses moving particle simulation (MPS), is now available with a wide range of features and improvements that complete the solver's ability to manage integrated physics (it includes a thermal solver), and make simulations faster due to its multi-resolution (MR) and multi-GPU (graphic processing unit) support.

Local particle-size refinement

One of the most exciting features of Particleworks 8 is the introduction of refinement zones. These allow the user to reduce the particle size (used to discretize the fluid being analysed) in specific areas of the domain, thus increasing the accuracy of the simulation without affecting calculation time.

Previously, the entire fluid domain (and the air when present) was discretized using a single particle size. Now it is possible to define specific regions in the domain (boxes, custom domains, or offsets of the geometric elements of the model) with smaller particle sizes.

This creates possibilities for more detailed analyses of the lubrication and cooling of bearings that feature small gaps or flows through an electric motor's windings. For example, particle refinement has been used to evaluate the flows in the channels of an electric motor's cooling system. With the MR function, the flow in a narrow channel is solved with a particle size of 0.1mm, while the rest of the fluid domain is discretized with particles of 0.3mm.

Advanced modelling of grease, snow, and chocolate

It is also worth mentioning the introduction of a solver for high-viscosity fluids, resulting in significant improvements to the modelling of chocolate, grease, and snow flows. To handle these flows, the basic MPS algorithm has been extended with a "full-implicit" approach (moving particle full implicit - MPFI), whereby pressure and viscosity equations are solved in a coupled and implicit manner, increasing the accuracy of the calculations.

An increasingly integrated solver: air and thermal simulations

This version also expands the possibilities of the multi-phase simulations built into the software. The finite volume solver for air (based on a one-dimensional Cartesian grid) has been enhanced with a variable discretization lattice Boltzmann solver for more detailed analyses of external aerodynamics and vehicle contamination (e.g. interaction with puddles, rain, or mud).

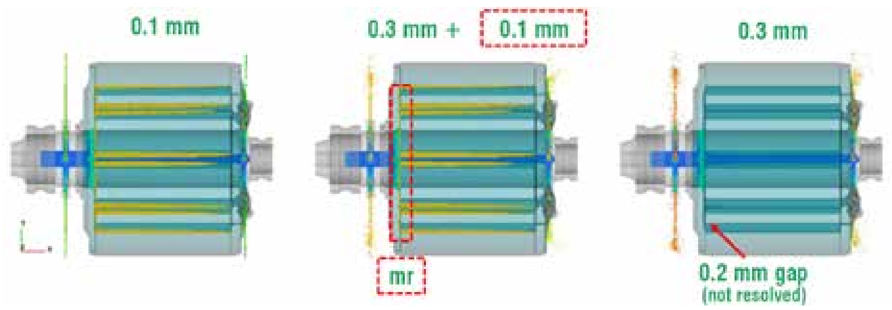


Fig. 1. Oil flow in the channels of an e-motor integrated in a rotor: by defining a multi-resolution region to resolve the small gap (dashed box), the particle size is only reduced locally, while the rest of the domain is discretized with a larger particle. This enables simulation times to be reduced by half.

Fig. 2. The MPFI solver improves the simulation results for highly viscous fluids. Particleworks 8 also includes advanced snow models for more comprehensive analysis describing different snow textures and degrees of wetting.

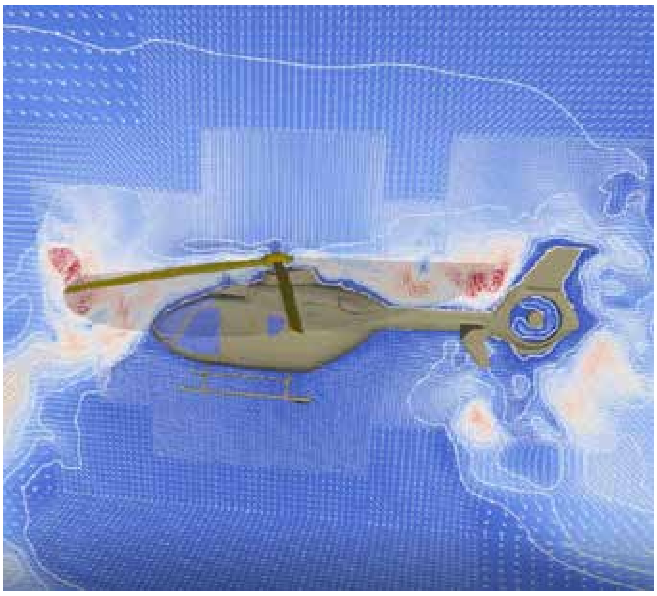


Fig. 3. Results from a lattice Boltzmann simulation of a helicopter take-off.

This mesh-less approach does not require the configuration of the calculation grid, simplifying external aerodynamic analysis without the need for geometric simplifications or for dealing with complex kinematics, such as the rotation of a helicopter rotor or the intricate dynamics of a damped vehicle.

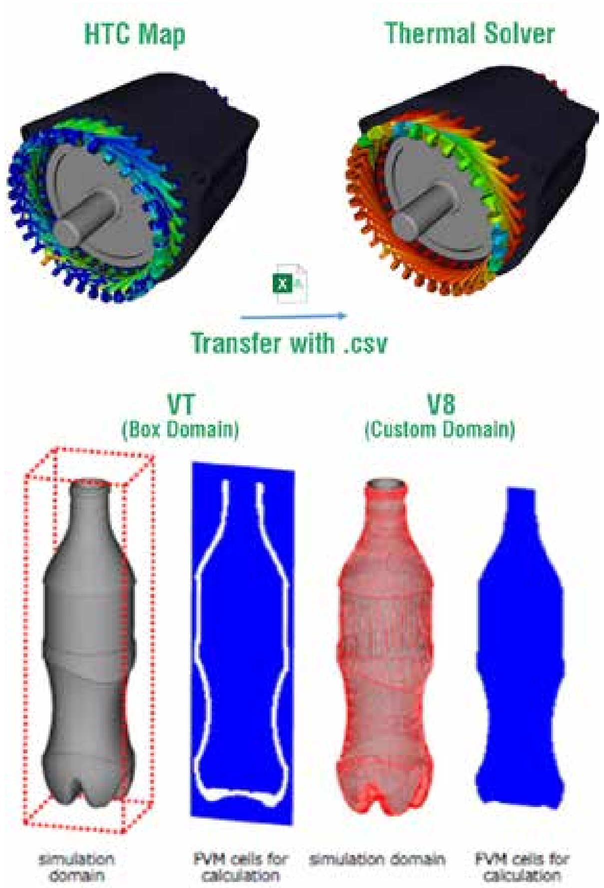


Fig. 4. Text files (.csv) can now be defined as boundary conditions for HTC (heat transfer coefficient) and temperature maps in the Particleworks thermal solver. Moreover, it is also possible to define customized simulation-domain shapes (.stl files).

More control over boundary conditions

As far as boundary conditions are concerned, the finite element thermal solver (already present in the previous version) now allows the user to set temperature and heat exchange boundary conditions on specific surfaces using .csv files.

The simulation domain can be defined with a customized file limiting the areas initialized with Cartesian grids for the air phase only to the area of interest (e.g. inside a bottle or engine compartment).

Efficient hardware scalability and other improvements

Compared to CPU core-based computing, running Particleworks on GPUs can provide significant advantages as shown in Fig. 5. It is worth noting that a single GPU running on a workstation can provide

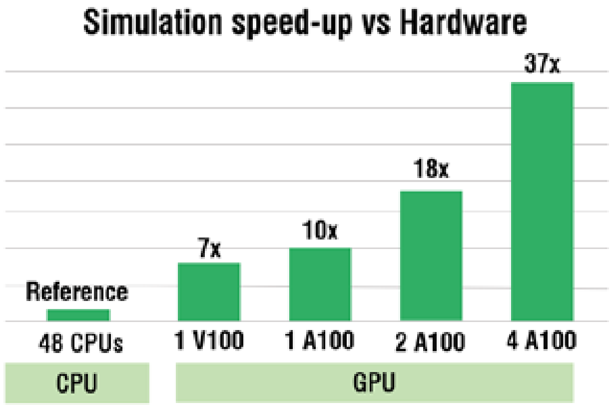


Fig. 5. Scalability analysis of Particleworks on single and multi-GPU, as compared to 48 CPUs. By running on 4 GPUs the simulation times could be reduced by a factor of approximately 40.

as much computing power as 450 CPU cores, at a significantly lower price and without the need for a large IT infrastructure. In addition, Particleworks also supports multi-GPU computing. The scalability is almost linear, with simulation times approximately halved when doubling the number of GPUs.

This new version includes other noteworthy improvements, especially for handling large-scale models, turbulence modelling, Python workspace integration into the GUI, and the introduction of passive scalar to evaluate chemical diffusion in the fluid/air.

In conclusion, the new release of Particleworks allows users to tackle more detailed and complete analyses faster and without sacrificing calculation time.

For more information:
Michele Merelli - EnginSoft
m.merelli@enginsoft.com

ENGINSOFT

SMACT competence center

18 Ottobre 2023

9.00 - 13.00

c/o SMACT Padova
Fiera di Padova
Via Niccolò Tommaseo 59

Digital Manufacturing Day

Scopri come migliorare la progettazione e gestione degli impianti ed ottimizzare la produzione!

ISCRIZIONE GRATUITA MA OBBLIGATORIA
www.enginsoft.com/it/italy-events/digital-manufacturing-day.html