ENGINSOFT

The Simulation Based Engineering & Sciences Magazine

Newsletter

Year 17 n°1

Spring 2020

Using personalized biomedical engineering to improve patient treatment

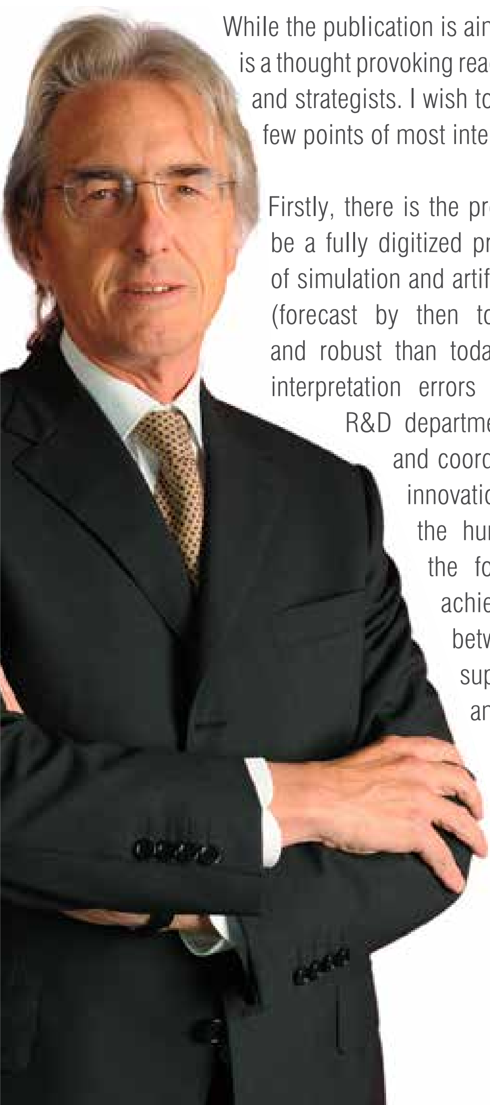
Introducing Skippy: an athletic monopedal robot, designed for a repertoire of behaviors

Improving design shape using adjoint solutions and surface morphing

Investigating wind-turbine structural behavior under icing conditions

Flash

The Fraunhofer-Gesellschaft is one of the leading organizations for applied research in Europe, with an annual research budget totaling 2.3 billion euros. Its public mandate is to play a central role within the German innovation system, in applied research and knowledge and technology transfer. It works to foster cooperation and institutional networking between science and industry. In mid-2017, it formed its first non-technological group, the Fraunhofer Group for Innovation Research, to focus on socio-economic and socio-technical research. At the end of 2018, the group published a document[1] called *"Understanding Change – Shaping the Future: Impulses for the Future of Innovation"* as food for thought for the future of innovation, which identifies five key opportunities for innovation by 2030 that also highlight related challenges to be addressed by industry and society today to succeed and thrive into the future.



While the publication is aimed at the German market, it is a thought provoking read for global business leaders and strategists. I wish to draw specific attention to a few points of most interest to those of us in SBES.

Firstly, there is the prediction that innovation will be a fully digitized process making massive use of simulation and artificial intelligence algorithms (forecast by then to be far more developed and robust than today and largely free of data interpretation errors and anomalies), requiring R&D departments to become integration and coordination centers for complex innovation networks and where the human contribution will take the form of radical innovations achieved by making connections between solutions, and supported by more simulation and testing.

For those of us immersed in the simulation-based sciences, this is good news and should stimulate us to greater efforts in the democratization of technology and the diffusion of SBES. Knowledge and experience in the use and application of these techniques will clearly become more sought after, while data management and data security will obviously also play a pivotal role here, as the Fraunhofer publication acknowledges.

Another key trend Fraunhofer identifies is that innovation will focus on integrated solutions in response to customer demand and will require the participation of customers, partners, suppliers, collaborators and other players. Due to the complexity of this, business models will need to be reengineered and the requirement to create value through innovation will drive essential reliance on a wide range of partner service providers. This represents a key opportunity for SBES professionals since the simulation of diverse business processes and alternative business models can provide fundamental support in this undertaking.

Digitally supported decision-making and creativity are clearly destined to become important keystones of the future of business and industry – particularly to address issues of sustainable development and meet the deep societal challenges that the passage of the coronavirus has thrown into stark evidence.

This issue of the magazine contains numerous articles that demonstrate various examples of digitally supported decision making, in product, technical and research-oriented environments. My hope is that it provides food for creative and strategic reflection at this critical moment in the world's history.

May we all evolve and succeed as we strive to collectively respond to the challenges of this hopefully soon-to-be post-Coronavirus world.

Stefano Odorizzi,
Editor in chief

[1] Fraunhofer Group for Innovation Research (Ed.) (2018): Understanding Change – Shaping the Future. Impulses for the Future of Innovation. Stuttgart. Available online at http://publica.fraunhofer.de/dokumente/N-509887.html

Contents

Case Studies

Interviews

Research & Technology Transfer

Software Update

Newsletter EnginSoft
Year 17 n°1 - Spring 2020

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

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

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

The EnginSoft Newsletter is a quarterly magazine published by EnginSoft SpA

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

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

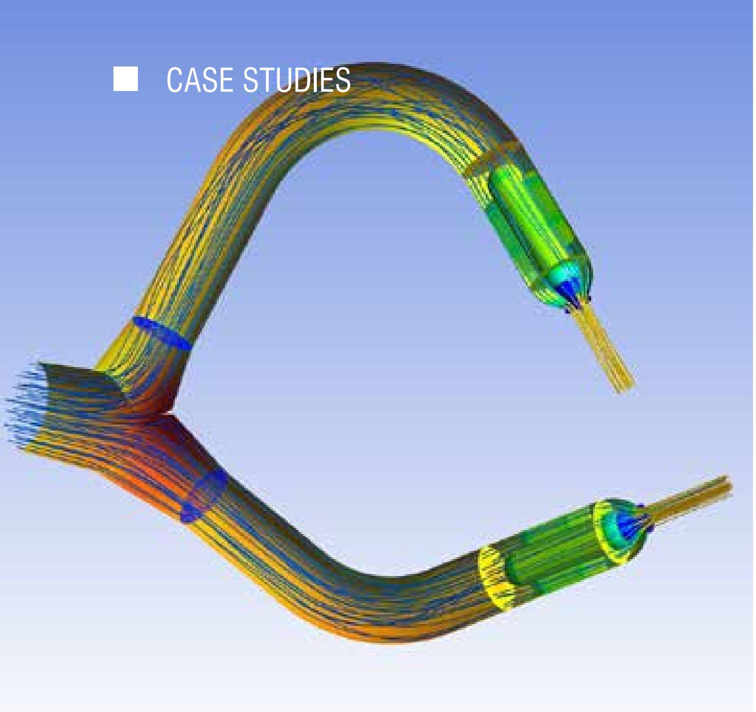
ASSOCIATION INTERESTS
NAFEMS International www.nafems.it • www.nafems.org
TechNet Alliance www.technet-alliance.com

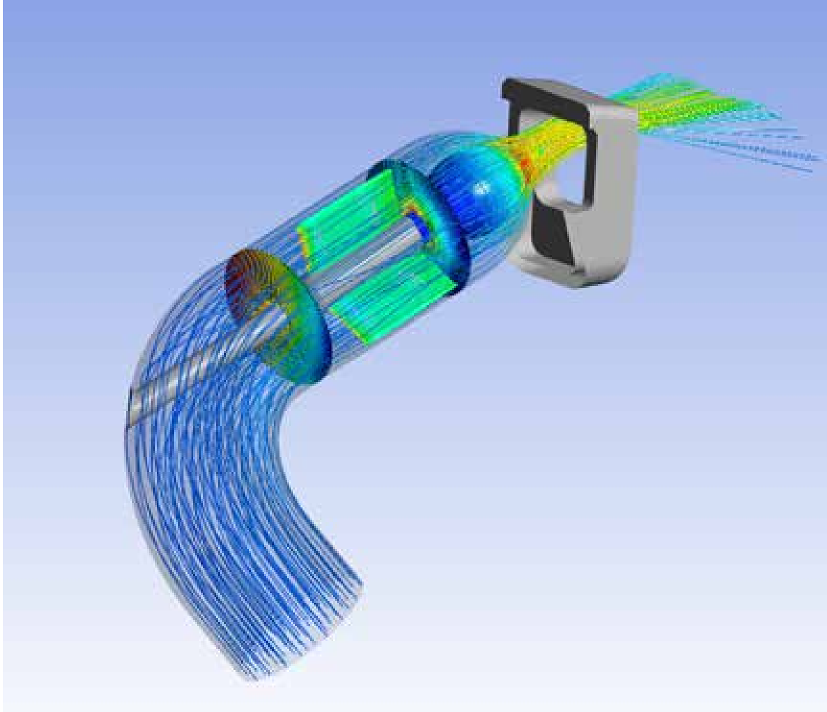
ADVERTISING
For advertising opportunities, please contact our Marketing office at: info@enginsoft.com

RESPONSIBLE DIRECTOR
Stefano Odorizzi

PRINTING
Grafiche Dalpiaz - Trento

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



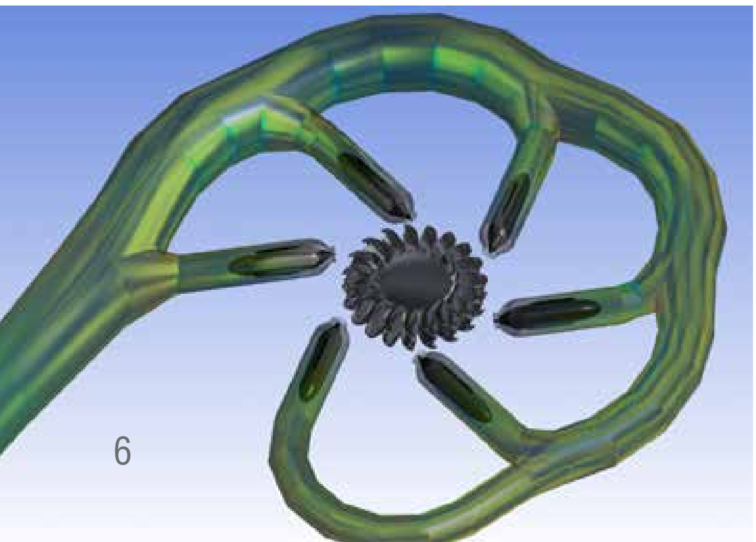


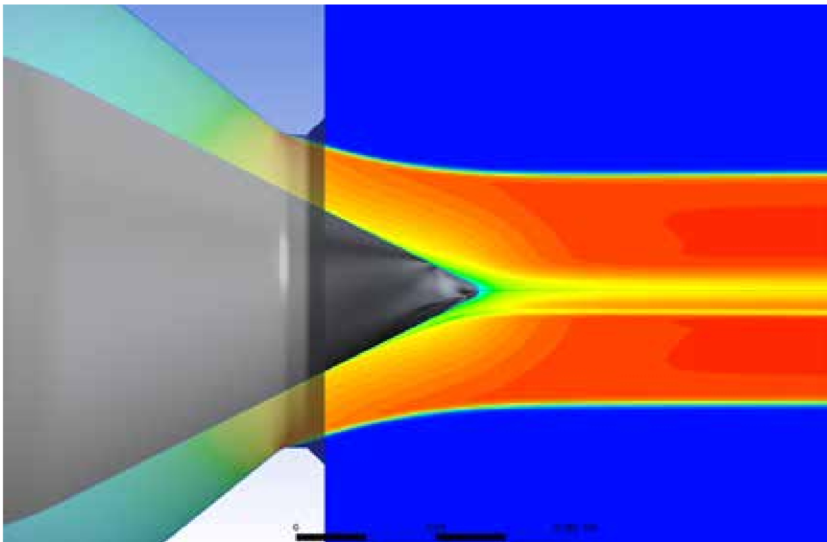
CFD study of a Pelton turbine runner

Comparison between traditional Eulerian and novel Lagrangian approaches

By M. Minozzo[1], R. Bergamin[1], M. Merelli[2], M. Galbiati[2]
1. ZECO - 2. EnginSoft

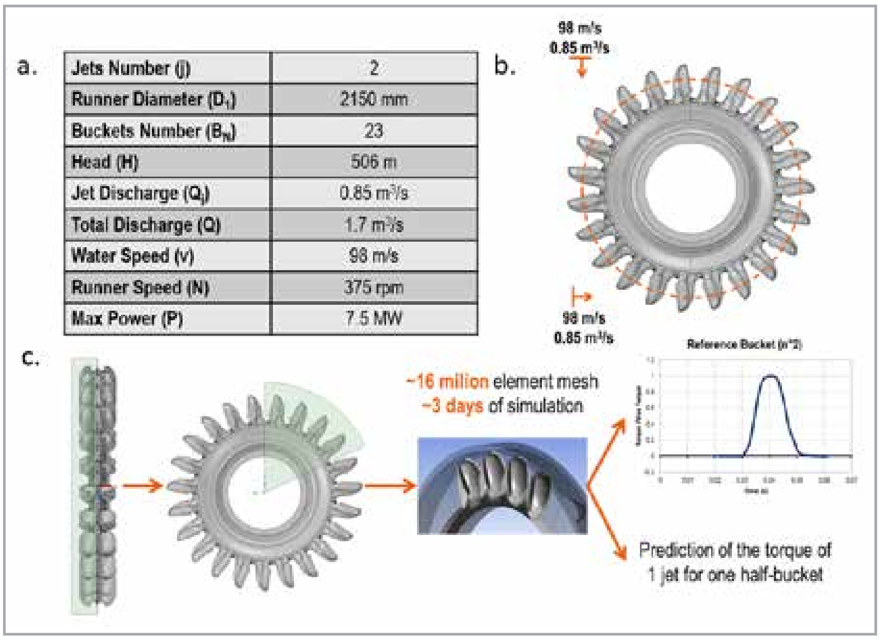
Hydroelectric power generation is currently the predominant source for low-carbon power generation and to support grid stability in the face of the growing use of other unpredictable, renewable energy sources. This means that water turbines are becoming the focus of increased study and optimization. This technical article, a collaboration between EnginSoft and ZECO, compares two different methodologies for the study and optimization of impulse turbines, specifically Pelton turbines, in order to evaluate which is the quicker and more reliable method. Pelton impulse turbines are more difficult and challenging to analyze than reaction turbines due to the complexity of their fluid dynamics and the resulting computational resources required for the necessary transient multiphase simulation. The unsustainably high time and computing requirements mean that there are some technical deficiencies in sector knowledge about specific elements of these turbines and their functioning, such as the inside of the water jet. Two methods were evaluated: the traditional Eulerian approach and a novel Lagrangian approach using Moving Particle Simulation (MPS). The novel MPS approach proved to save considerable time and revealed information not discovered before, opening up new possibilities for optimizing these turbines.





Hydroelectric power generation is a crucial source of electricity, accounting for 44% (IEA, 2020) of global low-carbon power generation. Its leading role is expected to be consolidated, as it becomes reinforced by developing countries and by the growing awareness of climate change. In addition, the renovation or repowering of old power plants is crucial for greener power production and to support grid stability, considering the growing use of unpredictable renewable energy sources, such as wind and solar.

The combination of these factors will increase the need to study and optimize water turbines under different conditions, not only at nominal design points. The standard Eulerian computational fluid dynamics (CFD) approach has been extensively tested and validated for reaction turbines such as Kaplan and Francis turbines.



Jets Number (j)	2
Runner Diameter (D_1)	2150 mm
Buckets Number (B_N)	23
Head (H)	506 m
Jet Discharge (Q_j)	0.85 m³/s
Total Discharge (Q)	1.7 m³/s
Water Speed (v)	98 m/s
Runner Speed (N)	375 rpm
Max Power (P)	7.5 MW

Fig. 1 - a) Summary table with details on the presented benchmark; b) Boundary conditions and geometry included in Particleworks (Lagrangian model), where no geometry modification was performed; c) Schematics of the geometry simplification necessary in CFX (Eulerian model) – the turbine is reduced using symmetry (sectors in green), the simulation is run, and the half-bucket profile is extracted.

It is already standard practice to optimize their hydraulic design due to the limited computing resources required. Impulse turbines, such as Pelton turbines, have also been continuously studied using CFD [1], [2], [3], [4]. These studies usually focus on predicting the efficiency of the buckets and on the fluid behavior of the water entering and leaving the individual buckets, in order to understand how the bucket geometry influences the performance of the machine.

However, compared to reaction turbines, Pelton analysis is much more complex and demanding, both because of the fluid-dynamic complexity of the jet diffusion, and the computational resources required for transient multiphase simulation.

The lack of knowledge about the inside of the water jet – as a result of the unsustainable time and computing resources required – is a technical deficiency that needs to be addressed. For these reasons, ZECO partnered with EnginSoft to investigate a new methodology to quickly and reliably conduct CFD simulations for Pelton turbines.

This article discusses the differences between a turbine runner simulation using a classic CFD (Eulerian) approach and a Moving Particle Simulation (MPS) (Lagrangian) approach. The test case presented is the analysis of a two jets horizontal shaft Pelton turbine. The project data is shown in Fig. 1b.

Conventional CFX (Eulerian) approach

From a hydraulic point of view, the Pelton turbine consists of a water inlet pipe or penstock, from 1 to 6 nozzles, and a runner. The manifold is a pipe, branched into up to six deviations, that leads water to the injector nozzle. The nozzle consists of a needle, which acts as an opening valve, and a water flow regulator that releases the flow in a free jet that impinges on the runner.

From a fluid-dynamic point of view, manifolds and nozzles are quite simple to study as they are either channeled flows or two-phase flows with a jet in the air in a limited and static portion of the volume [5], [3]. Instead, studies of runners involve greater challenges, due to the complex nature of the free-surface flow to be modeled. A Eulerian multiphase analysis of a complete turbine is time consuming and limited by the computational power requirements due to the complexity of the geometry and the simulation. To conduct a feasible Eulerian CFD analysis, the following assumptions establish the best practice for a traditional CFD simulation (see Fig. 1a):

- Reduction of the geometry using symmetry
- Reduction in the number of buckets analyzed, down to a minimum of three
- Creation of a domain (a statoric-rotoric for the inlet boundary condition and the rotating runner).

This enables the torque of a half bucket to be simulated and calculated for the full duration of the action of a single jet.

From there, it is necessary to work backwards to reconstruct the torque for the entire turbine. In other words, starting from the torque produced by a single jet in a half bucket, the torque must be doubled to calculate the torque of the whole bucket. The complete time history of the turbine runner's action is reconstructed manually to yield the total torque and its average value (see Fig. 3).

Using the planes of symmetry, it is possible to visually reconstruct the interaction of the jet with the bucket to better visualize the interaction between the two. This approach accurately estimates the power and therefore the performance of the machine and the hydraulic behavior of a bucket. However, it is obvious that some issues remain unresolved because some hypotheses do not always apply. In addition, jet-jet and jet-casing interactions are totally excluded from this CFD approach, as the simulations required to analyze these phenomena are unfeasible in an industrial R&D workflow.

Advantages of the MPS methodology

Particleworks uses a Moving Particle Simulation (MPS), a CFD approach in which the fluid is discretized into particles (computational fluid volumes). The Navier-Stokes equations are solved on these particles using a Lagrangian approach which does not require the mesh-generation step, as the fluid has already been discretized. This allows for rapid model preparation and poses no additional problems when moving/rotating domains or wall boundaries are considered.

Typically, software based on this methodology is widely used in the automotive industry, where gearboxes, electronic axles and transmissions are simulated in whole-simulation systems. Other types of applications are soiling, mixing tanks and cleaning-jet analysis. In fact, thanks to its Lagrangian approach, Particleworks is ideal for the study of complex, free-surface flows. In this article, we present another interesting possible application: using MPS to improve product properties and design.

As mentioned, preparing and reducing the geometry slows the simulation time and limits the amount of information that can be extracted from the simulation. On the contrary, thanks to the

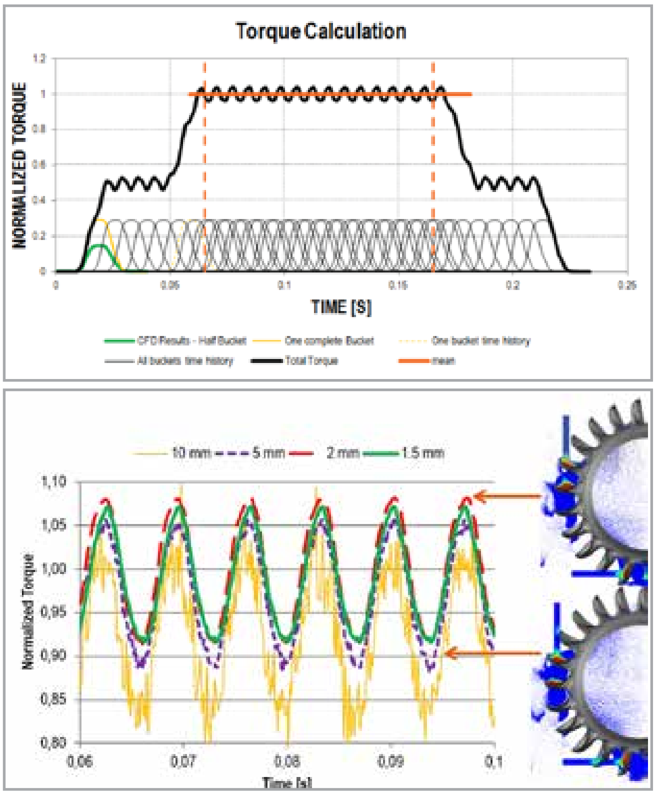


Fig. 2 - Top: Normalized torque predicted by CFX – the average value is obtained after reconstruction of the turbine profile from an initial half-bucket profile; Bottom: Normalized torque based on the configuration of the entire turbine – the minima and maxima can be related to specific jet-turbine interactions (on the right).

characteristics of the MPS method, the preparation phases and times are considerably reduced. In fact, the geometry provided by ZECO only needed to be converted to a compatible format for Particleworks (Fig. 1c). It was possible to import the entire turbine without the splitting or meshing steps. After setting the numerical and boundary conditions, the simulation was ready to run. The simulation process was further accelerated by the possibility of parallel processing, enabled by the graphics processing unit (GPU) solver. In addition, it can be seen that the extraction of the torque prediction was easier and did not require the time-consuming profile reconstruction steps.

Just like in conventional CFD, computed results improve with smaller mesh features, at the cost of longer simulation times. In general, you can observe a convergence for better, theoretically expected results. In Particleworks, this type of analysis is performed by changing the particle size, i.e. the dimension of the computational volume. In this way, a solution can be found independent of the simulation settings and the discretization of the fluid volume. We performed several simulations with particle sizes of 10, 5, and 2mm. To quantitatively analyze the results, we extracted the torque on the turbine and plotted it over time. As can be seen, the torque prediction graph becomes smoother and converges into values closer to the theoretical value (Fig. 2).

To further validate the simulation results obtained using Particleworks, we compared them to the CFX simulation results. As can be seen from Fig. 3, both software packages overestimated the overall efficiency of the Pelton runner by the same percentage. The difference between the two approaches is negligible and simulations within a 1% error margin can be considered an excellent result considering the literature in this field ([2], [6], [7]).

MPS not only achieves qualitatively comparable results to traditional CFD, it does so in less time. Because it can simulate the entire turbine, it also provides design information about long-range runner-water interactions. This makes it possible to analyze the effect of residual water in otherwise active buckets, or other undesirable interactions between the water and the turbine. In addition, the optimization of the casing can be accomplished with the same simulation.

Another type of analysis that is usually performed in this sector is the evaluation of the static mechanical stresses on the turbine buckets. In CFX, due to the division of the simulated domain, remapping the pressure from the data of only the half bucket is time consuming. On the other hand, due to Particleworks' integration with Ansys Workbench, data transfer to an FEM solver is simple(see Fig. 4c).

To summarize the comparison between the Particleworks (Lagrangian) and the CFX (Eulerian) approaches, the simulation steps and their related time-costs are presented in Table 1. As

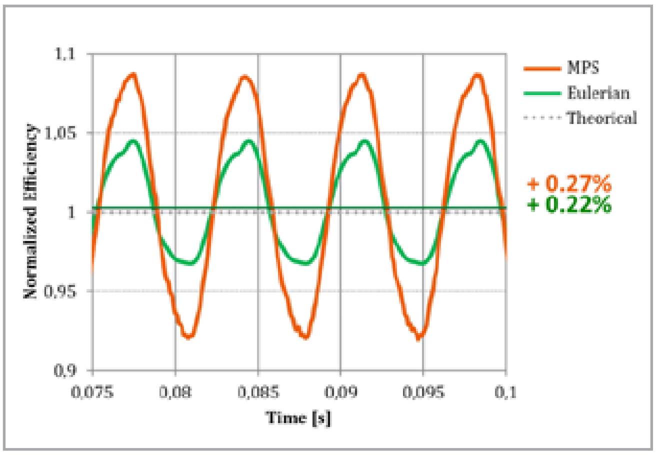


Fig. 3 - Normalized efficiency prediction for Particleworks (in orange) and CFX (in green). The percentage of error is reported at the side. The theoretical mean values are also reported (dashed, black line).

References

[1] K Patel, B Patel, M Yadav, and T Foggia. Development of Pelton turbine using numerical simulation. In IOP conference series: Earth and environmental science, volume 12, page 012048. IOP Publishing, 2010.

[2] Zh. Zhang. Pelton Turbines. 2016.

[3] D. Jo˘st, P. Me˘znar, and A. Lipej. Numerical prediction of Pelton turbine efficiency. 25th IAHR Symposium on Hydraulic Machinery and Systems, 2010.

[4] V. Gupta, V. Prasad, and R. Khare. Numerical simulation of six jet Pelton turbine model. Energy, 104:24–32, 2016.

[5] J. Zhang, Y. X. Xiao, J. Q. Wang, X. J. Zhou, M. Xia, C. J. Zeng, S. H. Wang, and Z. W. Wang. Optimal design of a Pelton turbine nozzle via 3D numerical simulation. Asian Working Group - IAHR's Symposium on Hydraulic Machinery and Systems, 2018.

[6] T. Kumashiro, H. Fukuhara, and K. Tani. Unsteady CFD simulation for bucket design optimization of Pelton turbine runner. 28th IAHR symposium on Hydraulic Machinery and Systems, 2016.

can be seen, Particleworks enables a significantly faster and easier simulation procedure. Since time is crucial in industrial applications, simulation times can be the bottle neck that block the development and investigation of new products. Various applications are not studied with CFD because of the complexity of the simulation steps. Particleworks can both accelerate the development of products that have already been studied, and pave the way for new studies and optimizations.

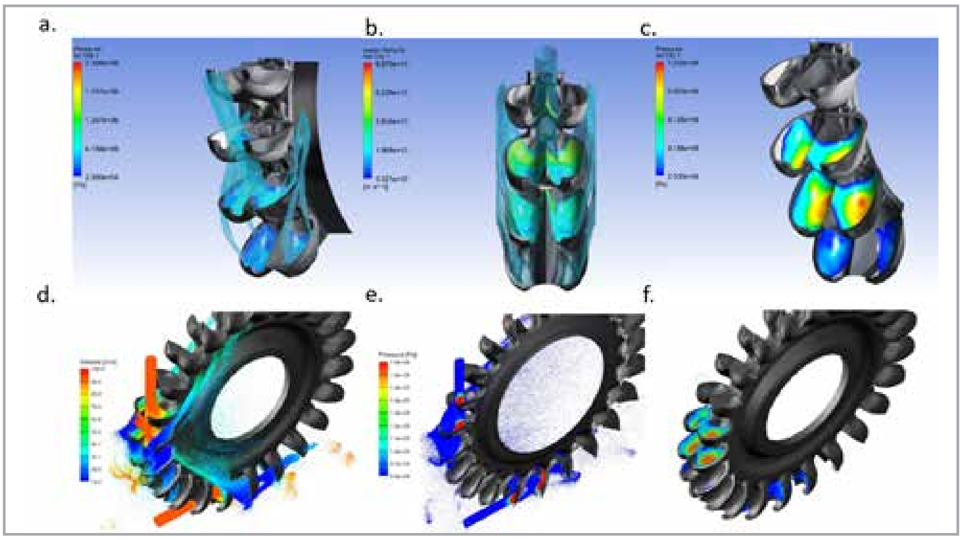


Fig. 4 - a) and b) Images of the reconstructed surface (using the mirror plane) of the water jet for CFX (Eulerian method) – the velocity and pressure profiles are mapped on the Pelton bucket; c) Reconstruction of the pressure profile on the runner bucket (Eulerian method); d) and e) Images of the two water jets simulated with Particleworks (Lagrangian method) – the color map represents the predicted velocity and pressure; f) Mapping of the turbine pressure profile – Ansys Workbench allows direct data transfer of the profile to the finite element method (FEM) solver.

Conclusions

This article has analyzed the outstanding issues with and the possibilities of simulating a Pelton turbine runner using CFD. The traditional Eulerian, mesh-based approach was compared to the MPS method.

We found that the qualitative results obtained are comparable and in good agreement with the theoretical values. The Eulerian approach, however, obtained this result through a complex definition and simplification of the model, requiring a considerable amount of simulation and working time.

	CFX	PARTICLEWORKS
Pre / Post Processing	3 working days / 4h	2h / 1h
Simulation time	70 h	2h
Simulated rotation (°)	138°	225°
Geometry	4 half buckets	Complete turbine
Complete runner simulation (multi jet, casing…)	Not feasible	Possible
Mesh elements/particles	16 M	4M
Hardware	12 CPU (Intel Xeon X5650 @2.67 GHz 96 GB RAM)	1 CPU + 1 GPU (NVIDIA V100)
Calculated vs model efficiency (absolute)	+0.22%	+0.27%

Table 1 - Summary comparison between the two approaches analyzed, highlighting working and simulation times, geometrical assumptions and hardware settings.

On the contrary, MPS can easily simulate the entire runner and the estimated workflow should only take 2-3 days. Moreover, the MPS method, from the same simulation, provides additional information never before investigated.

For instance, it provides insights into the jet-jet influence and the long-term jet-runner interactions. Those observables, together with the considerable acceleration in simulation time, open up new product optimization possibilities in the field of Pelton turbines.

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

Using personalized biomedical engineering to improve patient treatment

Two examples of the use of numerical methods to fine-tune personalization

By Christopher Blase, Achim Hegner, Armin Huss, Hans-Reiner Ludwig, and Andreas Wittek
Frankfurt University of Applied Sciences

This article describes two projects undertaken by the Personalized Biomedical Engineering Laboratory of Frankfurt University of Applied Sciences. The projects – the development of personalized prostheses, and the use of personalized diagnostics in medicine – are described in detail, showing the emphasis on individualized product and process development, which is the founding principle of the research laboratory. The laboratory focuses on research activities in each field in the development chain of personalized products that interact mechanically with the human body.

Personalization in biomechanics

Personalization or individualization is an important concept in medical treatment and product development and it has gained increasing attention over the last decade. The benefits of personalization are improved functionality and compatibility which lead to better performance, extended life and decreased burden on the individual's quality of life as well as on the health care system by focused treatment. In the development of prostheses as well as other technical devices which interact with human bodies, personalization is more and more important and required. At the University of Applied Sciences in Frankfurt, Germany (FRA-UAS) a research laboratory has been established in which the whole process chain for the development of personalized products and treatments is represented [1]:

Starting with the acquisition of relevant data such as acting loads, material properties and geometry (e.g., gait analysis and medical imaging) and the mechanical-mathematical modelling of material behavior (human hard and soft tissues), computer simulation methods are used to develop and improve artificial structures such as prostheses, which interact mechanically with the human body.

During development, the durability, material strength and biocompatibility of the resulting part are taken into account and verified by physical tests. After designing and optimizing the structures they can be machined or 3D-printed. In order to solve the tasks that arise from such a complex problem, eleven researchers work together in the aforementioned research laboratory, each with unique expertise in a specific field. Engineers as well as biologists and physicists work together in the laboratory and collaborate with clinicians and medical doctors to ensure the practical relevance of the results.The topics which are covered by this research laboratory are shown in two examples:

Example 1 deals with the development of an artificial partial knee replacement, example 2 with the diagnostic method and assessment of rupture risk of abdominal aortic aneurysms.

Partial replacement of cartilage in the knee

Clinical records show that implants for partial or total knee replacements must be replaced approximately 15 years after their initial implantation due to failure. Each replacement and revision makes it more difficult to achieve proper fixation in the bone. Therefore, the artificial joint increases with each replacement, resulting in a maximum of 2-3 revisions in total (Fig. 1).

Fig. 1 - Avoid total knee replacement for as long as possible

In young people with a defect in the cartilage of the knee joint, it is important to solve this problem with a minimum of bone- and tissue-replacement. Based on this idea, an implant for the localized replacement of the damaged cartilage only, with minimal damage to the bone, was developed as a first solution for these patients. This means the patients gain one further revision for knee joint treatment.

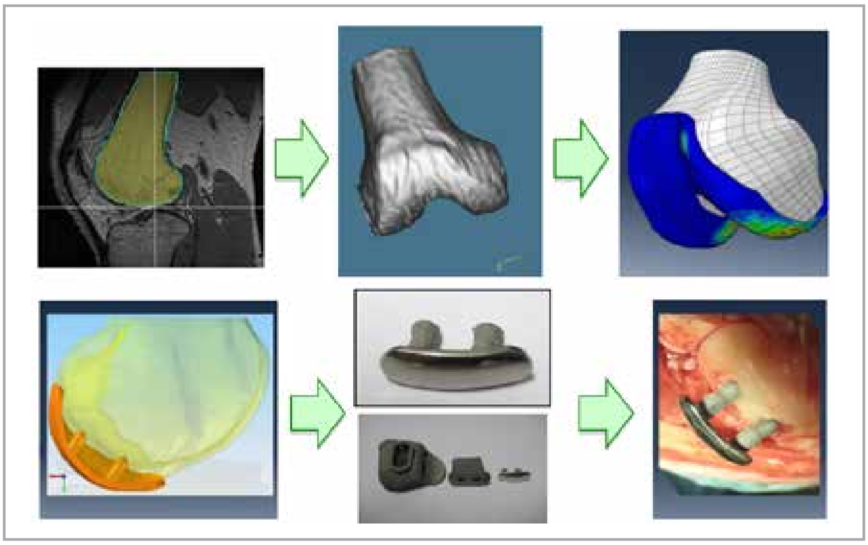


Fig. 2 - Development of partial prosthesis

To achieve this, the process chain shown in Fig. 2 was established. The individual biological geometry of the patient is recorded by extracting the 3D geometry from MRI images and transferring it to CAD data [2], [3]. Using this CAD data of the knee joint, an inlay is designed and manufactured to replace the damaged cartilage.

During the design process, Finite Element simulations are performed using the individual data to optimize the geometry and stiffness of the implant. The material properties are derived from tests of tissue material. Gait analysis is used to estimate the loads used for the simulation. After

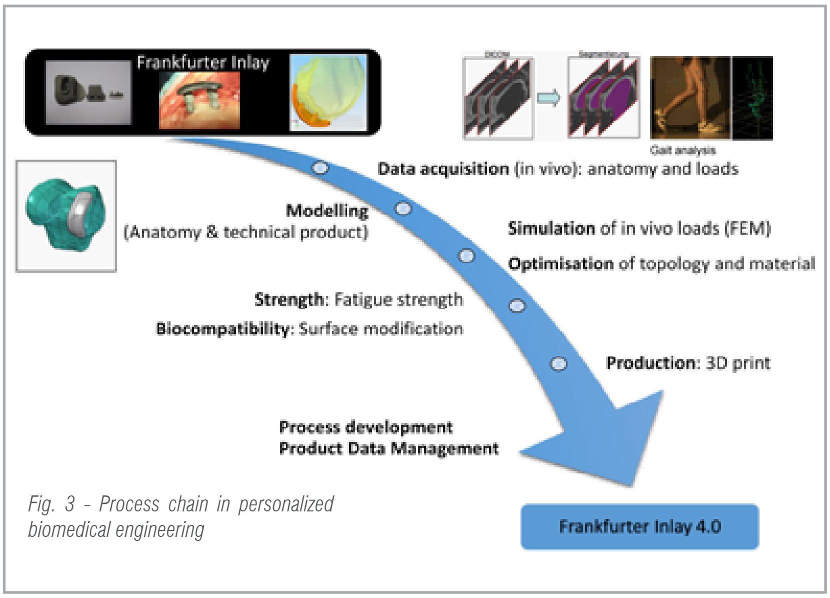


Fig. 3 - Process chain in personalized biomedical engineering

designing and optimizing the structure for the inlay, it is manufactured either by machining or 3D printing. The real inlay is then also tested for material strength, durability and biocompatibility.

Moreover, it is important to ensure a surface property that has very low friction on the joint-facing surface on the one hand, and a surface that allows the biological tissue to attach itself, to ensure a proper adhesion of the artificial material to the bone. In addition to the implant itself, special tools are manufactured that are used during surgery to ensure proper placement.

In an earlier project, a titanium cartilage replacement was tested in a clinical study on the knee joints of sheep. The inlay was designed to fit the individual geometry without doing simulations or the above-mentioned testing. Clinical studies showed that the inlay was probably much too stiff compared to the cartilage. This was one of the reasons for establishing a laboratory with experts from different disciplines. In a next step, the inlay will be developed according to the ideal process described above (and shown in Fig. 3).

Research activities are currently being carried out on the material characteristics of 3D-printed PEEK (polyether ether ketone). This development process for an artificial knee joint (Fig. 3) is an example of the development of personalized technical products such as prostheses and other products that interact with the human body (e.g., helmets or mattresses)

Investigations of Abdominal Aortic Aneurysms (AAA)

In addition to the development of these technical devices that interact with human bodies, a second example of the work in the research laboratory is shown: the development and improvement of treatment methods and diagnostic procedures. The example shown here is the improvement in the classification of rupture risk in aortic aneurysms. Such aneurysms are permanent aortic enlargements of more than 3cm. Abdominal aortic aneurysms

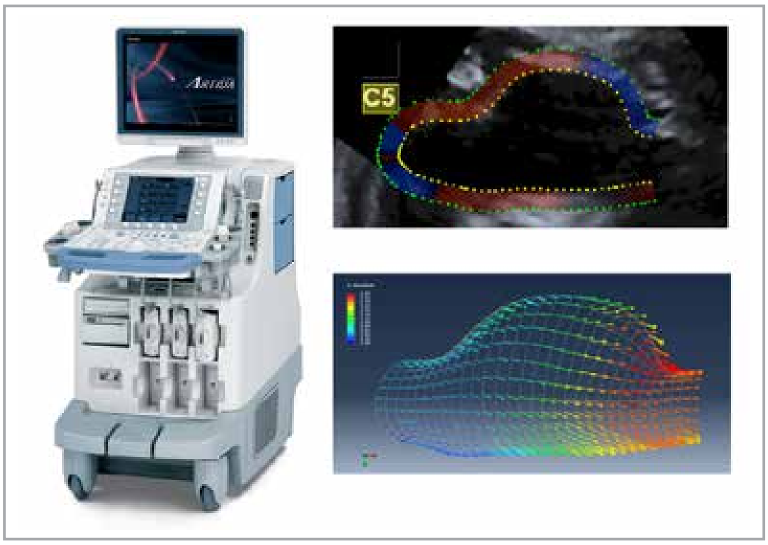


Fig. 5 - 4D-ultrasound-imaging and derived displacement-vectors of an AAA

(AAA) are highly dangerous due to the fact that after a rupture the mortality rate is about 50-80%. In over-65-year-olds, the prevalence is up to 9% among males and 2% among females. Until now, the criteria for surgery have simply been the maximum diameter of the aneurysm and its annual growth rate (Fig. 4). These criteria have been shown to correlate with the average rupture risk, but there are considerable individual variations. Surgery is performed when the diameter exceeds 5.0-5.5cm, or when the growth rate is greater than 0.5 cm/year.

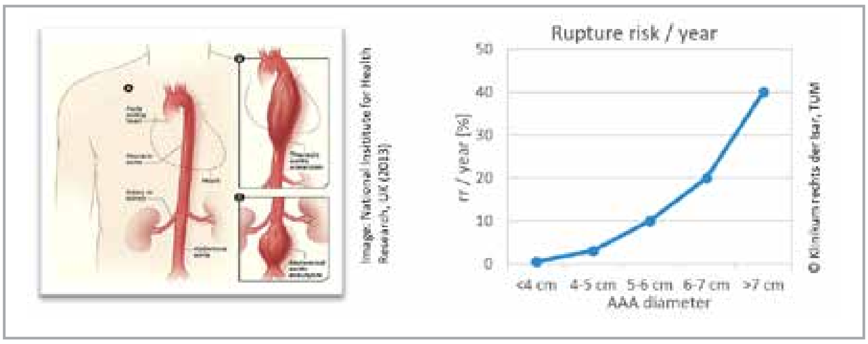


Fig. 4 - Rupture risk stratification of AAA

Neither criterion takes into account the individual, heterogeneous stresses or strains within the biological material that are the cause of the rupture (i.e. material failure of the blood vessel wall). Because of the generalized assumptions used, premature treatments are performed (AAA would be stable, even if above 5.5 cm), which entails unnecessary risks for patients and burdens for the healthcare system. Even worse, in the case of AAA with diameters of less than 5cm that are bound to rupture, any necessary treatment is delayed with fatal consequences.

The problem in this field is the lack of customization of the data on biological materials. The inter-patient variability of those data is quite high. Although it is often assumed that the tissue of young people is more flexible than that of the elderly people, this is not always the case. Therefore, material models are highly patient-specific and should be evaluated in vivo [4] [5] [6] [7].

The approach described in this example uses 4D-ultrasound data: three-dimensional ultrasound resolved over time. Using this data, the individual strains can be calculated (Fig. 5). By measuring the systolic and diastolic blood pressure, information on acting loads is also available.

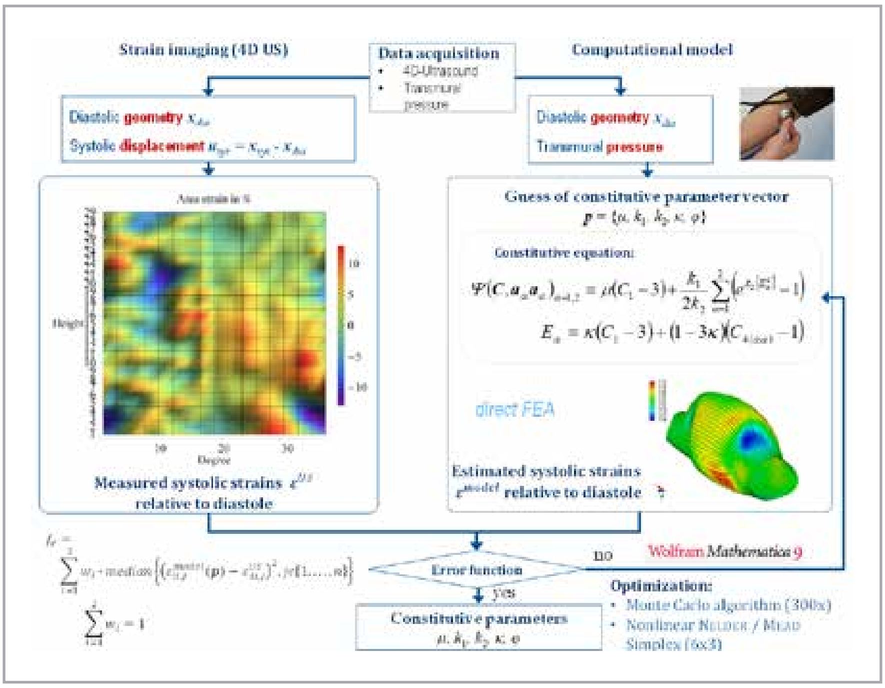


Fig. 6 - Derived time-resolved strains and constitutive parameter identification approach

Due to the fact that ultrasonic measurement provides complete data on field deformation, the law for the individual material can be identified by assuming nonlinear anisotropic elasticity. For the stratification of the risk of rupture, the individual stresses and strains in the AAA wall can be compared to limit values. (Fig. 6)

An even better method to evaluate the risk of rupture could be the assessment of the heterogeneity of the strain distribution [4] [5]. The advantage of this method is that no additional material models or assumptions would be needed and a pure strain approach could be used. Of course, strain limit values must be provided to evaluate the measured data.

This method is currently under clinical investigation. Another important extension is to achieve a higher spatial resolution and broader view of the measurements. Therefore, ultrasonic data resolved over time are correlated and recorded with CT data at a higher spatial resolution but with a lower resolution in the time domain [8].

Conclusions

The personalization of medical products and processes offers numerous advantages to patients and the healthcare system. Collecting individual data can be a challenge and may require the development or improvement of data acquisition. Integration of individual information can occur at several levels of the product development process and must involve experts from different disciplines.

The article describes and illustrates the work at the research laboratory with two examples. Obviously, Personalized Biomedical Engineering methods have far broader applicability than just these.

References

[1] Huß A.: "Personalized Biomedical Engineering - Examples of Using Numerical Methods for Better Treatments", 39th TechNet Alliance Meeting, 3.-4. May 2019, Rotterdam, NL.

[2] Betz S., Ludwig H.-R.: "Das „Frankfurter Inlay" – eine CAD/CAM-Anwendung für die Medizin", Der Ingenieurspiegel, August 2011

[3] Proschek D., Kafchitsas K., Kurth A., Mattvasovszky S.G., Betz S., Ludwig H.R.: "The quality of applied bone cement depends on the chemical composition of the application system", EurSurgRes. 2011; 47(4), October 2011, pp. 189-95 Epub

[4] Wittek A., Derwich W., Schmitz-Rixen T., Huß A., Blase C.: "Spatially resolved distensibility of aortic walls determined from 4D ultrasound measurements", 6th International Conference on Computational & Mathematical Biomedical Engineering, 10.-12. June 2019, Sendai, Japan.

[5] Wittek A., Derwich W., Fritzen C.-P., Schmitz-Rixen T., Blase C. Towards non-invasive in vivo characterization of the pathophysiological state and mechanical wall strength of the individual human AAA wall based on 4D ultrasound measurements. Z Angew Math Mech, 98, 2018, pp. 2275–2294.

[6] Wittek A., Fritzen C.-P., Huß A., Blase C.: "Inverse Identification of the individual, non-linear and orthotropic material properties of human internal anatomical structures based on non-invasive real-time 3D ultrasound full-field measurements". NAFEMS-Seminar: Material Properties in Structure Analysis - Modeling, Calibration, Simulation & Optimization. Dresden 2019. NAFEMS, pp 129–138.

[7] Wittek A., Derwich W., Karatolios K., Fritzen C.-P., Vogt S., Schmitz-Rixen T., Blase C. A finite element updating approach for identification of the anisotropic hyperelastic properties of normal and diseased aortic walls from 4D ultrasound strain imaging. J Mech Behav Biomed Mater, 58, 2016, pp. 122-138.

[8] Hegner A., Wittek A., Derwich W., Huß A., Schmitz-Rixen T., Blase C.: "A method for the registration of 3D blood vessel models based on computed tomography and ultrasound imaging", European Society for Vascular Surgery 7th Spring Meeting, 10. – 11. May 2019, Royal Free Hospital, London, UK.

For more information, email: huss@fb2.fra-uas.de or visit: www.frankfurt-university.de

Improving design shape using adjoint solutions and surface morphing

Testing a new workflow to save time and design iterations

By Simone Bartesaghi
Fluid4Engineering

This technical article describes the process undertaken by Fluid4Engineering to test a new adjunct method for CFD geometry modelling and morphing to save time and numbers of iterative models in the process of optimizing the CAD geometries for vehicle design for drag reduction.

In Industry 4.0, more and more investments are being directed to the generation of digital twin models for the optimization of industrial processes and factory design. Within industry 4.0, the reduction of C02 emissions is included in the concept of the green economy. An important part of the energy consumed in the world is related to road transport and most of the energy dispersion there is due to aerodynamic drag, which plays an important role in vehicle efficiency.

It is important to have tools that can effectively predict the loss of vehicle efficiency due to drag. In recent decades, wind tunnel experiments and computational fluid dynamics (CFD) have played a key role in predicting the vehicle performance of a generic bluff body. The usual design process consists of testing an initial shape (geometry) provided by the designer and using CFD data and visualizations to modify it in a close iterative loop between the designer and CFD engineers.

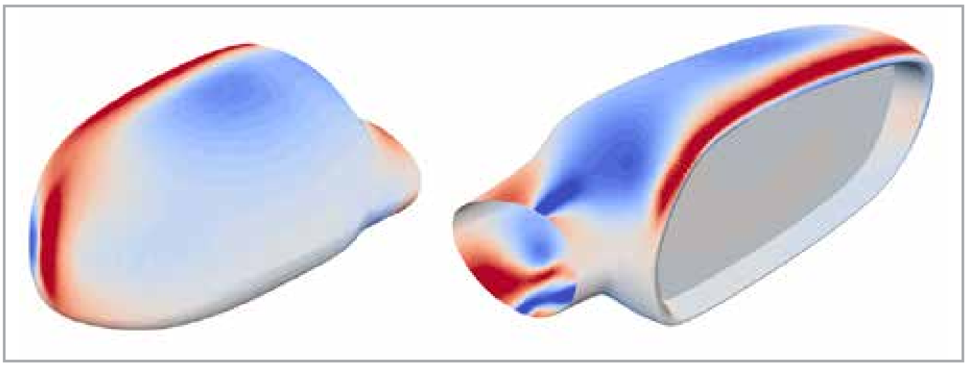


Fig. 1 - 3D wall-sense map of the aerodynamic drag-cost function

Cutting-edge simulation methods have been developed as a result of today's increased computational capabilities. In particular, new algorithms are being combined with classical CFD methods to calculate the effect of a shape in terms of its sensitivity for specific integral quantity targets (drag, lift, pressure-drop); adjoint equations are added to the digital model for the calculation of gradient values with respect to the quantity of interest.

The adjoint method predicts the influence of boundary conditions on objective function (cost). In general, the adjoint solution is calculated using a fully converged classical CFD solution called "primal"; using the information provided by the adjoint solver, it is possible to directly optimize the geometry without preparing different cases like in a classical design of experiments (DOE). The main advantage is the possibility to explore a potentially infinite number of designs and save time because, by working directly on a specific target, fewer design iterations are necessary.

In order to evaluate a possible adjoint method workflow to be used in production for geometry modelling and optimization, we tested a simple automotive-derived test case: the CAD for an isolated automotive rear mirror. The goal of the study was to modify the initial geometry to reduce the total drag by using the information from the CFD and adjoint solutions.

The adjoint solution provided the surface sensitivity for the chosen target function: aerodynamic drag. Using this information, the full 3D map of the CAD wall's sensitivity to aerodynamic drag is available and it is possible to modify the initial CAD according to this rule: BLUE pull normally outward (in the fluid); RED push normally inward (from the fluid), (see Fig. 1).
A direct morphing technique is used to modify the geometry; a discrete box of points is created around the CAD model and the correct displacements (point-per-point) derived from the 3D wall-sense map provided by the adjoint solver are applied. The magnitude of the displacement produced by the morphing method, which moves the discrete points around the CAD model, is shown in Fig. 2.

From the baseline CAD reference, the morphing algorithm generates the new CAD according to the wall-sense map; for the first iteration, the constraint imposed for the maximum deformation amplitude was 6mm. Fig. 3 shows the vectors (direction and magnitude) of the displacement of each point made using the morphing technique.

Continuing the classical CFD simulation on the modified geometry, the resulting drag reduction was in the order of 3% in a single design change step. The drag reduction is visible in the wake turbulence reduction, Fig. 4.

The encouraging results of this method led to it being applied to a more complex geometry: a simplified racecar. Once again, the target function was to reduce the aerodynamic drag. Using the same methodology and a 20mm constraint as the maximum displacement, the modified geometry was again tested using CFD and the results were compared to the baseline data. After post-

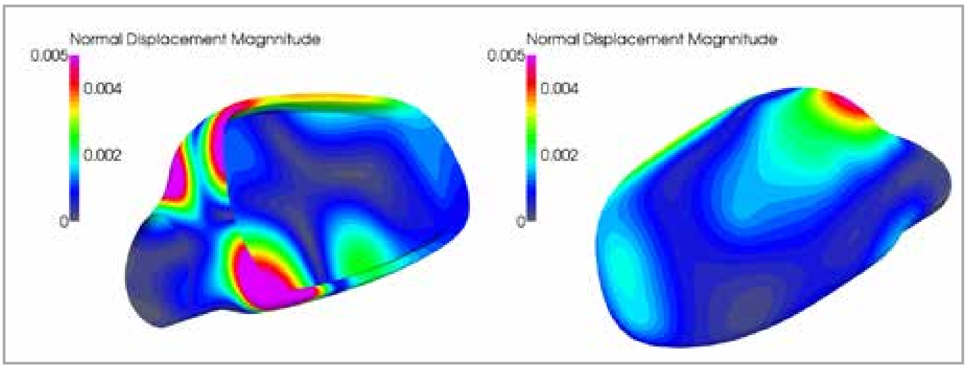


Fig. 2 - Normal surface displacement.

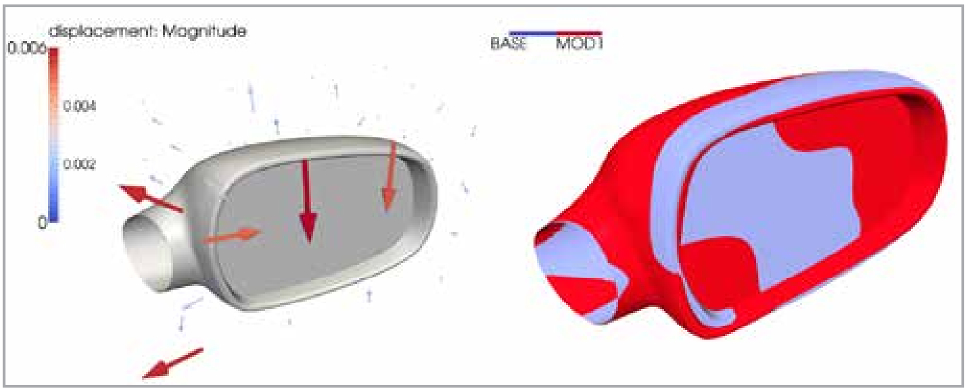


Fig. 3 - Discrete point with morphing direction vectors (left) and resulting surface (right).

About Fluid4Engineering

Fluid4Engineering srls (F4E) is a company started after the 35th America's Cup experience. Professionals involved in the F4E project have a curriculum enriched by participations in Volvo Ocean Race and America's Cup campaigns. In a way, F4E provides methods and tools useful to estimate fluid dynamics properties, mainly for sailing yacht and mega yachts; F4E develops also specific tools for optimization and design checks by using open-source technologies (mainly Python).

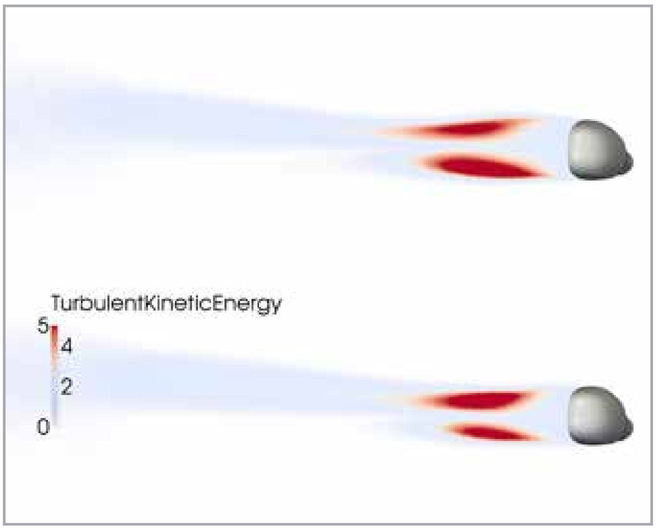


Fig. 4 - Turbulent kinetic energy distribution

processing the data, the drag reduction achieved was 3.5%; it must be specified that we were considering drag alone, without considering the lift of the car necessary to maintain ground stability.

Further investigations are required and planned. Figs. 5-8 show the drag wall-sense, the displacement magnitude and the comparison between the baseline and modified CAD surfaces.

Based on the results of this initial study, it can be assumed that wall-sense maps are useful for designers to define the most influential features of the CAD compared to the target cost function; in addition, other target cost functions, such as Drag Coefficient (CD), Lift Coefficient (CL), Pressure Drop, etc. could be used to further define the wall-sense maps and guide the CAD design using more information in order to improve the geometry optimization even more. This method is effective for reducing drag and noise (wake control), increasing the efficiency of bluff bodies and for guiding external aerodynamic design using fluid flow and effective objectives.

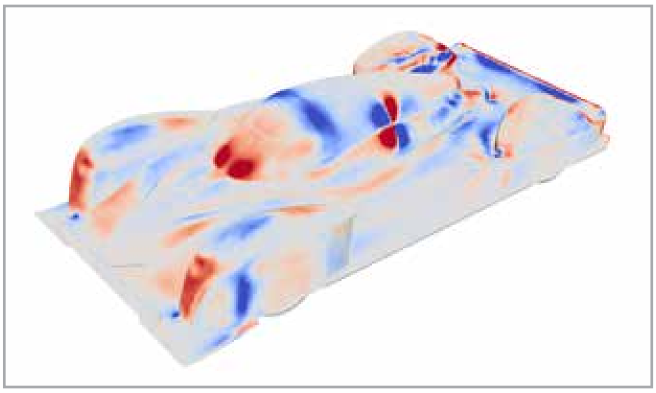


Fig. 5 - Wall-sense drag distribution

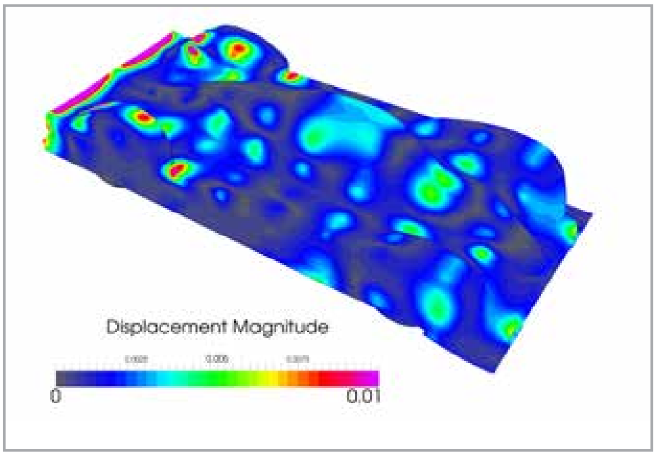


Fig. 6 - Normal surface displacement

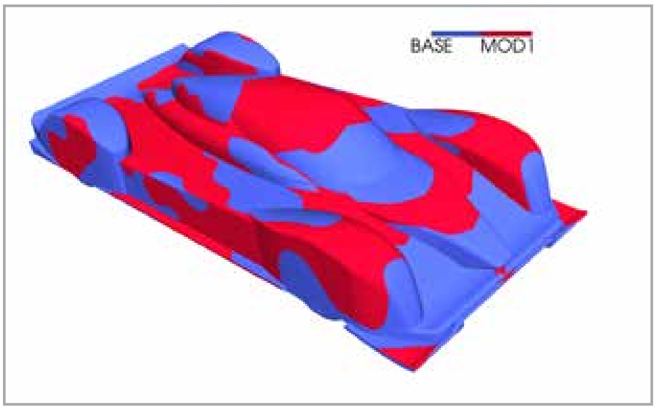


Fig. 7 - Baseline vs modified CAD surfaces

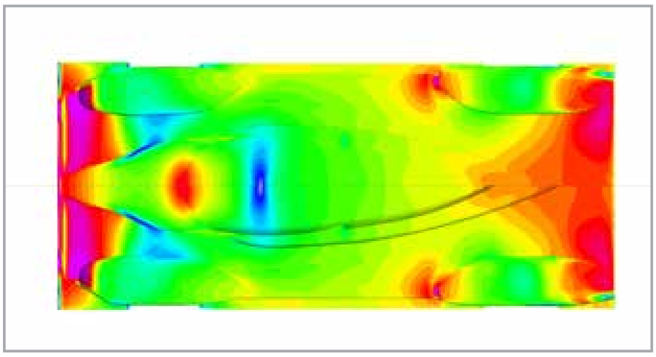


Fig. 8 - Pressure contour comparison: Baseline CAD (top); Modified CAD (bottom)

For more information visit:
www.fluid4engineering.com

For more information:
info@enginsoft.com

Innovative and effective handling and transport of sensitive equipment for the civil and defense industries

By Antonio Lagreca
Powerflex

Powerflex was founded in 1996 to design, manufacture and certify innovative turnkey solutions to safely manage and transport critical equipment in the most challenging operating environments.
The company's current key services are:

- Voice analysis for customers
- Preparation of a project management plan based on the scope, quality, time and cost of the project
- Development of virtual prototypes based on a design approach guided by structural and thermal finite element analysis (FEA)
- Development of a physical prototype to reduce risks and resolve design and environmental certification testing issues
- Environmental certification of equipment at its own environmental laboratory
- Manufacture of plug&play ICT cabinets, each supplied with a smart power distribution unit, a thermal-electrical cabinet management unit and interconnection cables
- Management of large manufacturing batches of metallic and glass-fiber reinforced plastic (GRP) containers in its 2000m2 plant

To maintain and improve its high standards, Powerflex's engineering department collaborates with the most important Italian universities and research centers. Current research projects in which the company is involved include Horizon2020, Clean Sky 2, Castle, POR (Programma Operativo Regionale) GRP/EMI shielding, and PCTO (Percorsi per Competenze Trasversali e per l'Orientamento) with the Giordani Institute & Co.

Powerflex's main market sectors are:

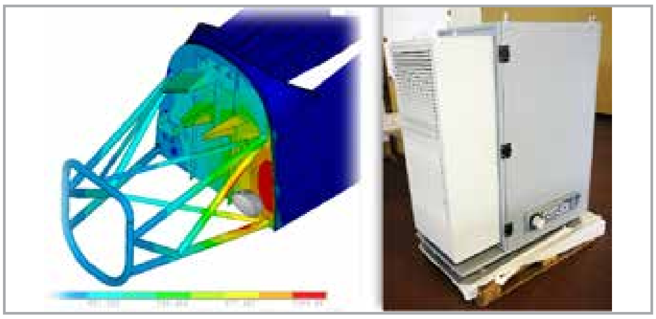
- Defense (Sea-Land-Air)
- Aerospace
- Rail
- Seismic
- Industrial

Some of the most recent projects it has completed, include:

- Structural and thermal FEA
- Containers for heavy weight torpedoes
- 85" shock-proof video wall
- Air conditioned shock proof cabinet with cabinet management unit

TECHNICAL CHALLENGES

Powerflex performs advanced structural and thermal dynamic FEA to support the design and development phases of its products and to reduce the manufacture of physical prototypes. The transition to Ansys represents a technical and strategic turning point for the company which previously carried out design and simulation activities using other simulation tools. "MSC NASTRAN code is no longer the only standard in the defense sector," states Antonio Lagreca, manager of the engineering department at Powerflex, "Ansys is rapidly gaining market share because it provides more flexibility, more functionality and great robustness.



The platform also allows you to automatically read and reconstruct all models in MSC NASTRAN, Abaqus, and Dyna formats, thus allowing you to use everything that has been done in the past with other software. Undoubtedly Ansys technology is gaining more and more popularity, especially for those involved in structural and Multiphysics analysis," he continues. Currently Powerflex has only one resource involved in simulation activities, but it intends to extend the use of Ansys software to new users. The main analyses undertaken are structural and thermal, static and dynamic, linear and nonlinear with a focus on the vibrational and frequential response of structures under the effect of stationary and transient loads over time. In this regard, the company plans to undertake further investigations in the near future in order to implement all the features of the Ansys explicit solver.

"EnginSoft has proven to be a reliable and reactive partner in supporting us in managing and manipulating the dynamic super element created in the MSC NASTRAN code. The wide availability and applicability of Ansys primarily allows us to adopt a simplified and organic data exchange format with customers and suppliers," explains Lagreca. *"It will also allow us to consider other markets and industrial sectors that could represent new opportunities for business growth."*

For further information visit: www.powerflex

For more information:
Fabio Rossetti - EnginSoft
f.rossetti@enginsoft.com

Introducing Skippy: an athletic monopedal robot, designed for a repertoire of behaviors

Demonstrating the effectiveness of a systematic, complete design approach to achieve extreme, unprecedented behaviors

ISTITUTO ITALIANO DI TECNOLOGIA

By Antonios E. Gkikakis and Roy Featherstone
Istituto Italiano di Tecnologia (IIT)

This article discusses a realistic multi-objective parameter optimization study of a highly athletic one-legged robot, called Skippy, in which both the parameters of the mechanism and the parameters of its optimal behaviors were sought.
The result is a Pareto front of robot designs that meet or surpass a set of behavioral performance objectives and, for each design, the set of command signals that accomplish these behaviors. The aim of the study was to identify a design that could meet all of the objectives, and then to build a real robot based on the optimal design. The study, which was restricted to planar motions (two dimensions), is highly realistic and detailed.

This article presents a parameter optimization study for the design of a highly athletic monopedal robot, known as Skippy, that is physically capable of performing vertical jumps up to 3m (4m in later prototypes) and triple somersaults of 2m, and which is able to stand and balance on one leg. The robot's design uses a recently invented transmission mechanism called a ring screw (http://www.royfeatherstone.org/ringscrew/index.html), which is largely responsible for its high performance. The use of the ring screw, an alternative to the widely used ball screw, allows the robot to achieve higher speeds and greater efficiency.

Even though Skippy has no immediate economic purpose, it serves to demonstrate the effectiveness of a more systematic and complete design approach in achieving extreme and unprecedented behaviors. While this case study focuses on a one-legged robot, its findings can be generalized to other types of robots with a properly defined purpose or set of objectives.

The main challenge of this study was that the performance objectives were set very high and the robot already operates near the limit of what is physically possible using today's technology. The objective of the study was to identify a single design with the highest overall merit from among all the designs that met all of the desired objectives. Achieving such an objective is generally not an easy task because objectives may often conflict with each other. For example, Skippy-like robots that can jump very high are usually not very skillful at balancing, and vice versa. High jumps require a lot of energy, which can be stored and re-used via springs; balancing, on the other hand, needs a stiff body, making these two objectives conflicting. For this reason, we chose a multi-objective optimization approach for our study.

But what is the point of a one-legged hopping robot? By studying a highly simplified monopedal robot, we wanted to gain a deeper understanding of legged locomotion, which we could then extend to multiple legged versions, and use the mobility offered by legged locomotion by taking advantage of the symmetry that characterizes it (think of a human's legs while running).

The framework of our study consists of a two-layer optimization scheme. In the first layer, an optimization algorithm searches the design space for the most athletic design. In the second layer, each design is tested for its ability to meet each of the behavior objectives. Each behavior objective is treated as a separate optimization problem. We define the term 'behavior' to describe what the robot does and the term 'performance objective' to describe the outcome of this behavior. An example of a performance objective, therefore, is for Skippy to landing after a 2m vertical jump and then launch itself back into the air to perform a 2m triple backward somersault. The conditions at the moment of landing, plus each of the robot's actions until the moment it lifts off from the ground, describe a complete behavior. A score that reflects how close each behavior is to the performance objective is then awarded for each behavior. This is an example of machine and behavior co-optimization.

The model and simulation

To achieve high performance, highly detailed and realistic models of the mechanism, its limits and inefficiencies are required. For example, the motor is subject to speed, torque and thermal limits, and there are energy losses in the motor, transmission, and springs, as well as when the robot's foot hits the ground.

Robot Skippy's mechanism was modeled with 56 parameters (the detailed listing of the parameters has been omitted due to its large size). The schematic diagram is presented in Fig. 1.

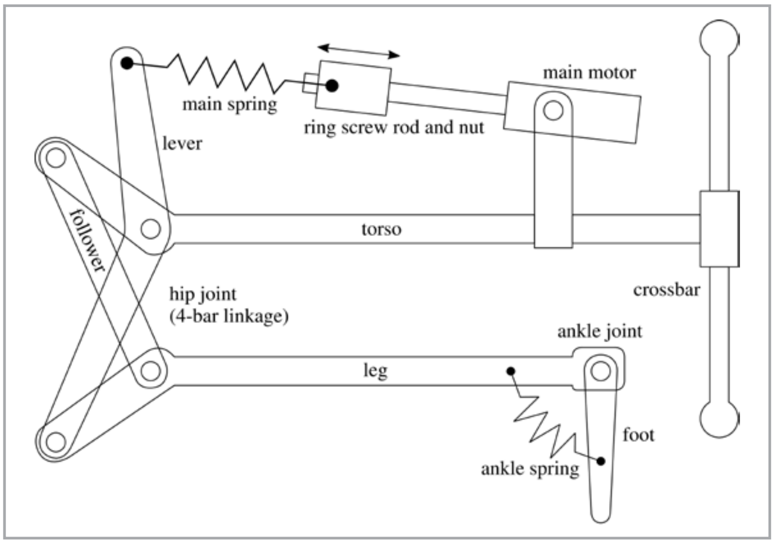


Fig 1 - A schematic diagram illustrating Skippy's relevant parts

The robot has three joints: the ankle, the foot and the hip. The ankle is a passive joint (it is not directly controlled) and has a spring attached to it. A second spring is attached in series with the ring screw nut, which is also connected to the main motor of the robot. This motor effectively controls the hip via a 4-bar linkage; both springs are custom made from glass fiber. Finally, Skippy

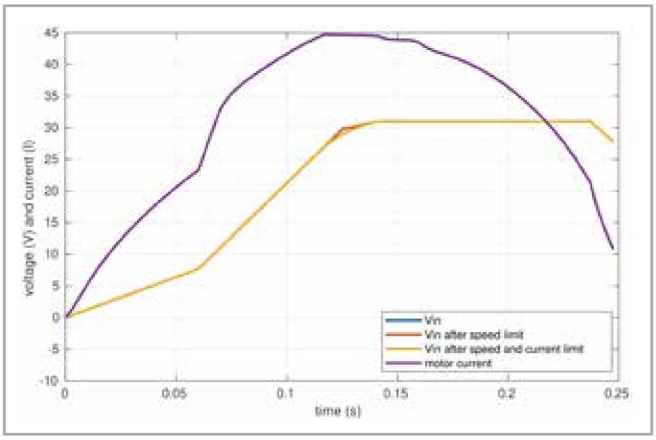


Fig. 2 - Voltage and current profile during a 0.247 second stance phase that results in a 2m triple backward somersault

has a crossbar (controlled by a second motor), which will be used to achieve 3D balance and steering, which are not relevant to this study.

We decided that some parameters would be fixed while others were the result of separate optimization studies (such as the 4-bar linkage). For the study presented in this article, we optimized seven parameters: six of these define the profiles of the two springs while the seventh defines the position of the center of mass of the upper link, which has a significant impact on the robot's performance.

Finally, we developed a simulation environment in which the mechanism's performance for each of the desired performance objectives was tested. A behavior was defined with 26 parameters of which 13 were independent variables representing the voltage profile fed to the robot's motor. Fig. 2 shows an example of a voltage profile. The simulation was performed with MATLAB and Simulink.

Optimization architecture and algorithms

Fig. 3 illustrates the overall organization of the process has been arranged into modeFRONTIER, platform for process automation, optimization and data mining, which, as mentioned previously, is organized in two layers. The upper layer finds a Pareto front of optimal designs that meet all of the objectives. The MOGA-II algorithm was used because it allows multiple objectives and because it is not limited to local exploration, such as gradient-based algorithms. The latter was important because we were managing a high dimensional non-linear search space.

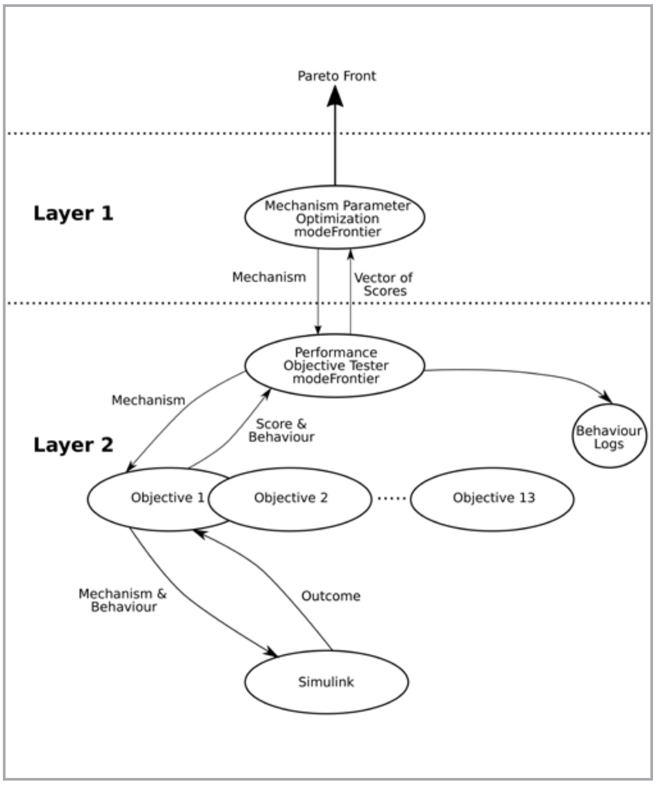


Fig. 3 - Overall organization of the design optimization process.

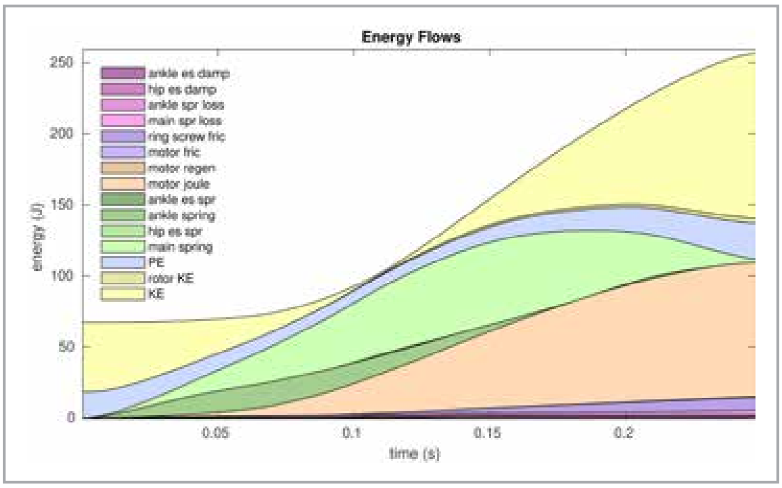


Fig. 4 - Stacked energy flow of 0.247 second stance phase that results in a 2m triple backward somersault.

The lower layer consists of 12 individual optimization experiments to find a given mechanism's best performance for each performance objective, and a calculation to determine that mechanism's balancing ability, which we wanted to maximize. To summarize, these performance objectives were to:

- perform jumps increasing in height (with no angular momentum or horizontal velocity) up to 3m from a resting phase
- return to rest via a series of jumps of decreasing height
- initiate, stop or continue performing travelling hops (with non-zero horizontal velocity and zero angular momentum)
- perform a 2m triple backward somersault after a 2m vertical jump
- maximize the physical balancing ability.

MOGA-II was also selected for this stage, with some of the performance objectives being formulated as multi-objective optimization problems.

Finally, the algorithms were subject to 11 constraints to ensure the realism and feasibility of the simulations. To name just a few, these were to: avoid penetrating the ground, avoid self-collision, limit the applied current and more. An overview of the algorithms, which were used in their default configurations, and their parameters is presented in the following table (Table 1).

Parameter name	1st layer (mechanism)	2nd layer (behavior)
Initial population	ISF + User-defined	ISF + User-defined
Population size	10	5
Number of generations	26	40
Algorithm type	Generational evolution	Generational evolution

Table 1 - Algorithms and parameters used

Design of experiments

The Incremental Space Filler technique and a user-defined initial seed were used to explore the search space. The initial seed is a mechanism which can nearly perform the desired objectives and their corresponding behaviors (it was identified via a separate optimization study using the same framework).

Results

The optimization scheme evaluated 260 designs, of which 46 (~ 18%) passed all the physical performance tests. A design was considered to be a successful candidate when at least one valid behavior per objective was discovered, allowing for a small interval of error.

We discovered mechanisms that were very skillful in one or in some of the objectives, but that performed poorly in others. The mechanism that could jump the highest under-performed in the traveling hops, while the mechanism that excelled in somersaults had difficulties in performing low hops. A new, more extensive experiment, to be conducted in future, will seek a deeper understanding of the physical traits that lead to this natural inclination for specific behaviors.

In this study, our main objective was to identify a single machine with the highest overall merit: a machine that displays high athleticism in all of the performance objectives, is a capable balancer, and expends the least amount of energy while performing these behaviors (this was an extra selection criterion that was not considered for the optimization). By examining the Pareto front solutions, we identified a design that satisfied all of the above criteria and this is currently being manufactured to test the effectiveness of our method.

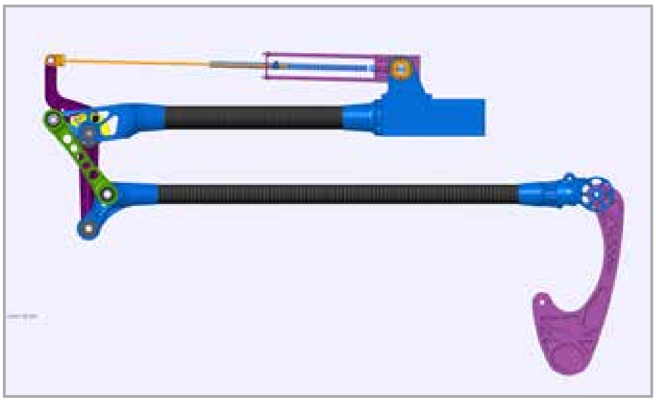


Fig. 5 - CAD design of Skippy. Real Skippy is currently being manufactured. Springs, ring screw nut and electric motor are not displayed.

Fig. 4 presents an sample behavior of our selected design and shows the stacked energy flows during one of the most demanding behaviors (the stance phase of a 2m triple backward somersault). The potential and kinetic energies can be seen as well as energy losses due to friction in the springs, the motor and the ring screw.

In Fig. 2, we presented the voltage and current profile of the same stance phase of a 2m triple backward somersault. Notice that the motor is in saturation (at 31 Volts) for a large part of the stance phase, proving that the mechanism is indeed being driven to its limits.

About the Istituto Italiano di Tecnologia (IIT)

The Istituto Italiano di Tecnologia (IIT), based in Genoa, Italy is a foundation financed by the State to conduct scientific research in the public interest, for the purpose of technological development. The IIT aims to promote excellence in basic and applied research and to promote the development of the national economy. As at December 2019, the IIT had produced more than 13000 publications, participated in over 200 European projects and more than 40 European Research Council (ERC) projects, made more than 900 active patent applications, created 22 established start-ups and has more than 40 more under due diligence. It has a network of 4 hubs in Genoa that form its Central Research Laboratories, a further 11 research centres around Italy, and two outstations located in the US at MIT and Harvard.

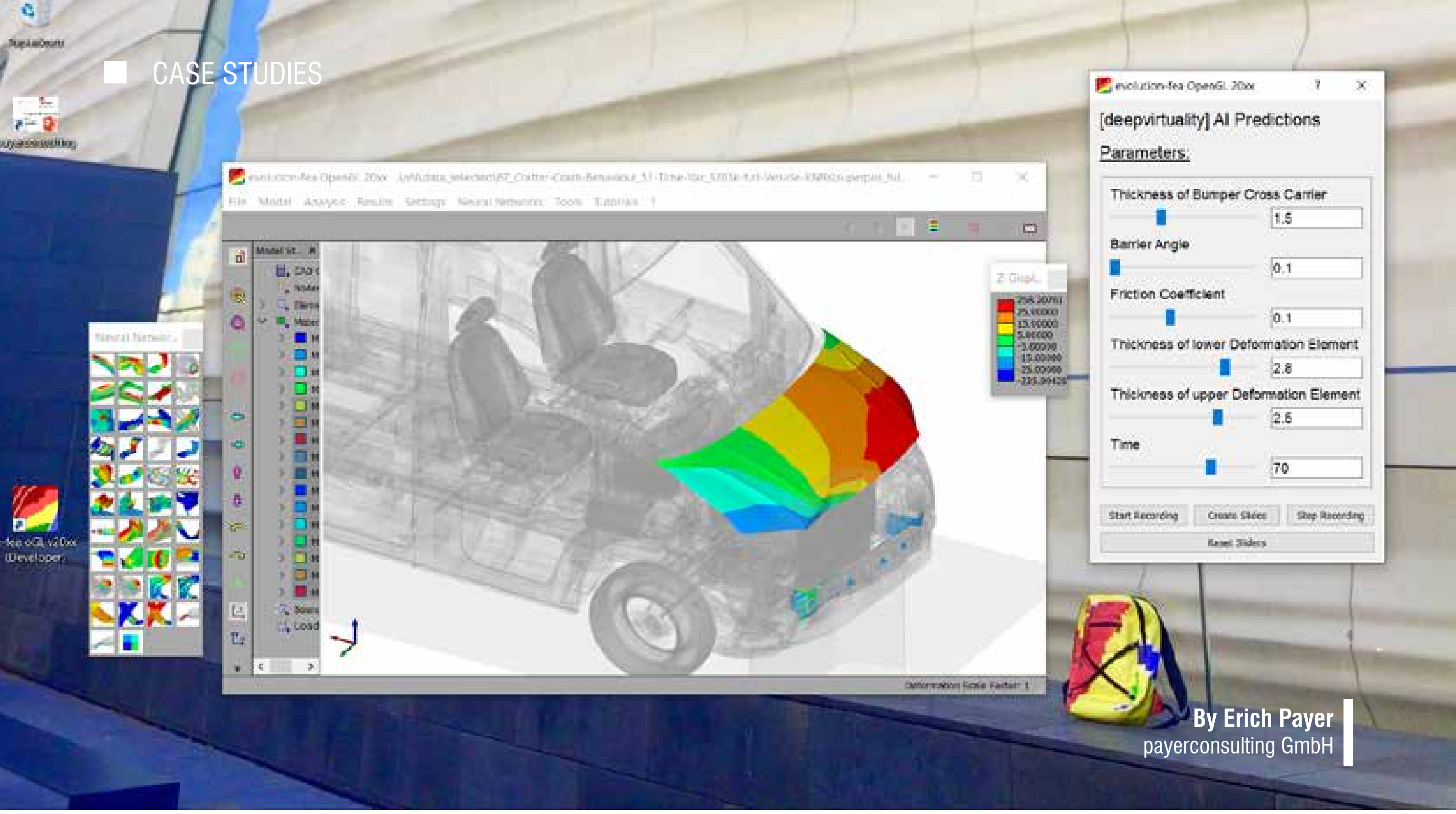
Conclusion

This study has demonstrated an optimization method for the design of a robot that meets several highly athletic behavior objectives. The study focused on the design of a highly athletic mono pod that can perform jumps of increasing height up to 3m, then jumps of decreasing height, traveling hops, display a high physical ability to balance and was selected based on an energy efficiency criterion. The framework evaluated 260 different designs of which a single design was selected to be built.

The study was performed in one plane, but is highly realistic and uses very detailed models and simulations that capture all the important energy flows. Despite the focus on detail, there are limitations to our work. We did not simulate our designs on uneven surfaces and we assumed a constant friction coefficient with the ground. We plan on investigating these issues, as well as on varying and introducing additional parameters to our model.

For further information about this project,
visit: royfeatherstone.org/skippy/

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



By Erich Payer
payerconsulting GmbH

AI saves VW 600 hours of crash simulation time

Neural Networks trigger a new era in product development

The use of artificial neural networks has greatly increased over the last couple of years and offers fascinating, new possibilities in many fields of technology. This brief technical article describes how payerconsulting, in collaboration with VW Data:Lab, developed a set of neural Network solutions for AI predictions of the key mechanical properties of products, such as stiffness, durability, vibration, acoustics, etc. for use in the area of industrial design and engineering.

These AI solutions and software tools comprise algorithms for rapid data generation/processing and easy-to-use graphical user interface (GUI) functions for nN specification, training, evaluation and deployment.

The use of simulation techniques today is standard fare in product development and enables designers and engineers to (basically) reduce and/or avoid the most time consuming and costly processes of physical prototyping. However, simulation models are usually built from 3D CAD models which necessarily requires the availability of such CAD surface or solid models. Since these CAD models always have to be processed to meet simulation needs (e.g. closing of gaps or removal of obsolete small fillet radii etc.), the creation of simulation models, particularly for subsequent analyses and result evaluations can be a very costly and lengthy process.

A process that is becoming lengthier, too, due to the steadily growing complexity and refinement of the finite element (FE) meshes. For example, a full vehicle crash simulation of a single design [or impact] variant on today's "typical" high-performance computing (HPC) systems takes up to 12 hours, or even more, of CPU time.

Artificial intelligence (AI) and neural Networks (nN) for product development

On the other hand, the use of artificial neural networks has greatly increased over the last couple of years and offers fascinating, new possibilities in many fields of technology. The key advantage of nN is their ability to 'learn'. They, thus, represent a software-

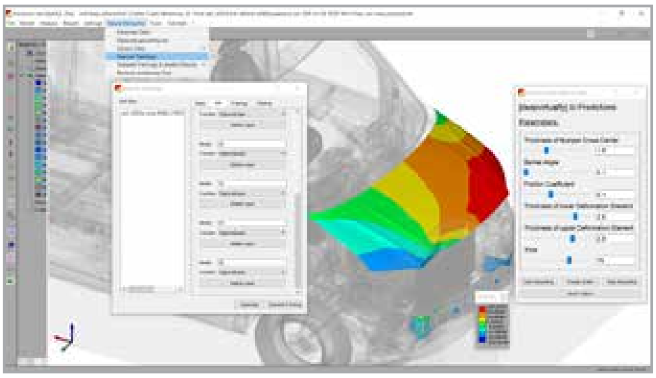


Fig. 1 - e-fea toolbox for rapid nN generation and deployment

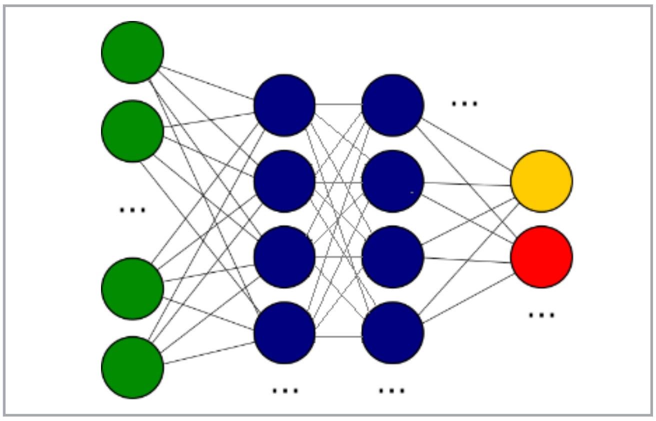


Fig. 2 - nN topology – input (in green), hidden (in blue) and output (in yellow and red) layers

based system that is similar to the actual human brain: they are able to determine relationships and recognize patterns; they can be 'trained' on existing data sets; and they are already being successfully used for text, speech or image recognition, autonomous driving issues, and more.

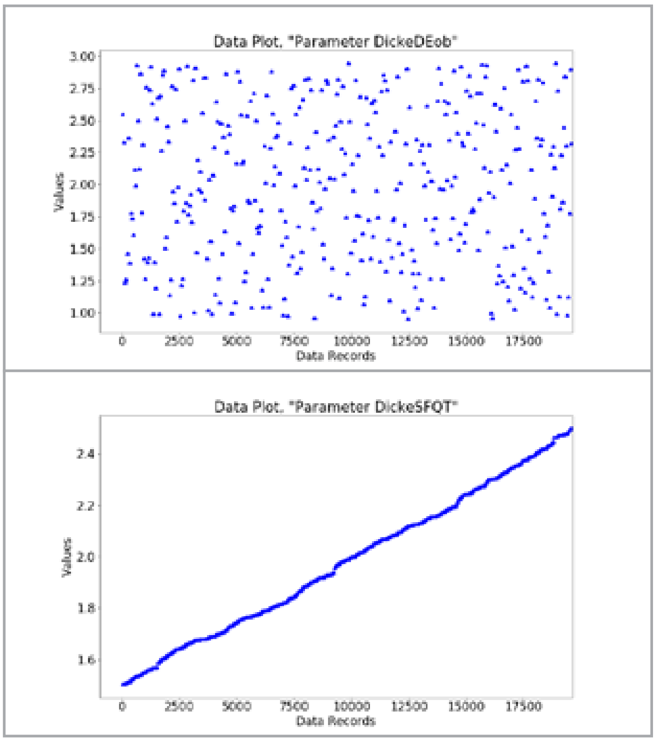


Fig. 3 - nN training data – input parameters e.g. thicknesses of upper crashbox elements and bumper cross carrier

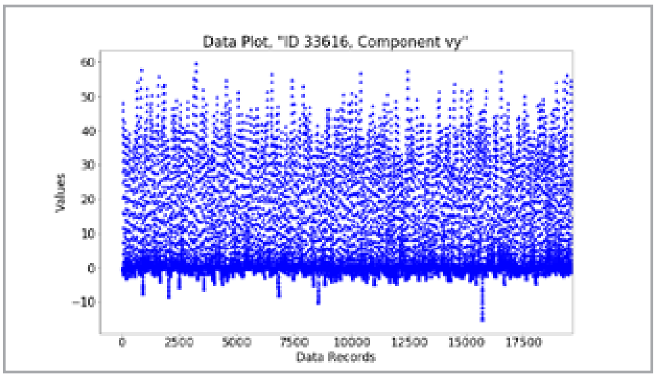


Fig. 4 - nN training data – output values e.g. displacements of a crashbox FE node

However, to date, in the area of industrial design and engineering, nN solutions using AI predictions of the key mechanical properties of products, such as stiffness, durability, vibration, acoustics, etc. have not yet been available.

To address this and in close collaboration with the VW Data:Lab, payerconsulting has been developing a set of tailored AI solutions and software tools based on eOSSP's open source FE kernel, evolution-fea (e-fea), and on the open source AI library, PyTorch. These AI solutions and software tools comprise algorithms for rapid data generation/processing and easy-to-use graphical user interface (GUI) functions for nN specification, training, evaluation and deployment (see Fig. 1).

For instance, to generate a nN for crash predictions of the VW Crafter (i.e. a Feedforward nN for regression with a basic topology similar to Fig. 2), we used existing Pamcrash FE analysis (FEA) results (4TB of data) covering a design of experiments (DOE) matrix with 400 statistically-determined variations/combinations of the relevant design and impact parameters, and an FE model of the van (with 3.2 million FE nodes and elements) for GUI-based parameter variations and direct visualization of the nN predictions.

The six nN input parameters were: the thicknesses of the upper (#1) and lower (#2) crashbox elements, the thickness of the bumper cross carrier (#3), the impact angle (#4), the friction coefficient (#5) of the contact area, and time (#6), while the computed FE displacements were used as associated nN output values (9.6 million for the full vehicle).

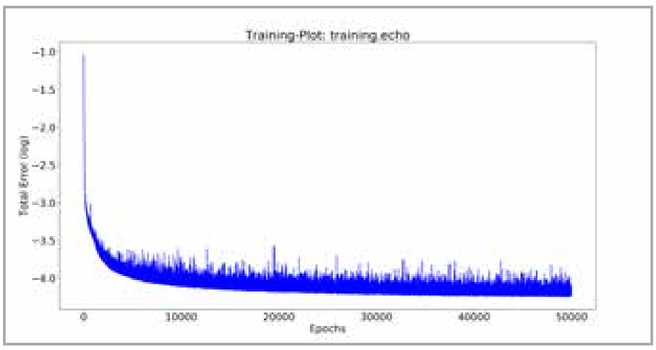


Fig. 5 - nN Training history – error vs. epochs

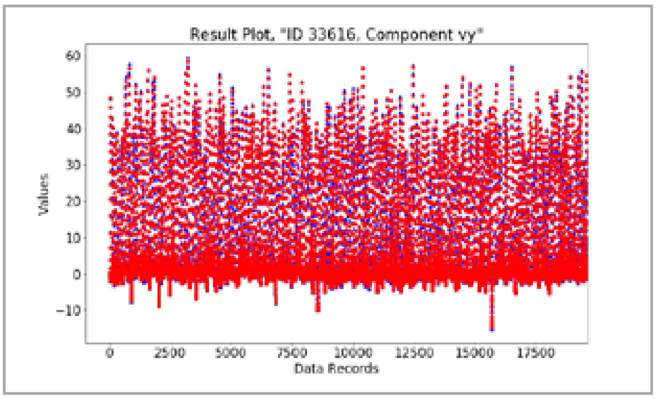


Fig. 6 - Training data validation – nN predictions (in red) vs. target data (in blue, see Fig. 4)

Based on these data sets (see excerpts in Figs. 3 and 4), and using the functionality of our nN Toolbox (i.e. dialog boxes for nN specifications such as number of hidden layers and nodes, type of activation function, learning method etc.; GPU/Cuda options for short training times, comprehensive evaluation and plotting features, etc.), we were easily able to generate the nN to perform highly sophisticated, live VW Crafter crash predictions (see Figs. 5 and 6).

Furthermore, the validation of the test data showed excellent correlations between the nN predictions and the re-checks of the FEA (see Fig. 7), while saving VW 600 hours of time in crash simulations.

> *"That's why AI makes a good engineer an even better one!"* — *Volkswagen AG*

About payerconsulting

Founded in 1992, payerconsulting today is a widely acknowledged engineering consultancy, developer of breakthrough AI solutions and tailored simulation tools, and strategic R&D partner of numerous market leaders from different industrial areas.

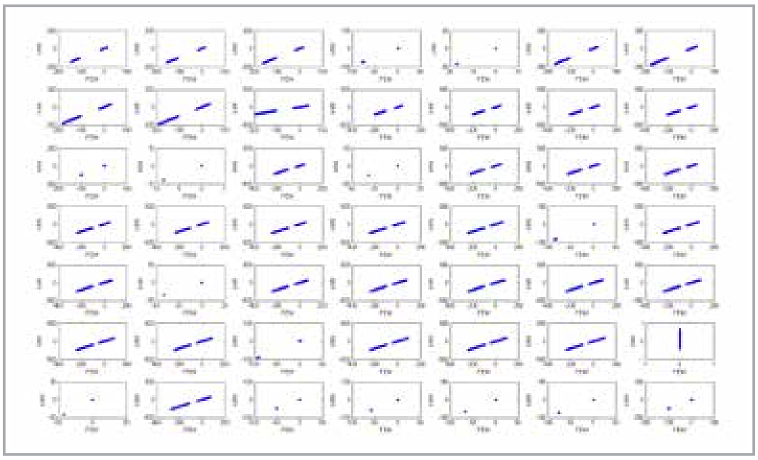


Fig. 7 - Test data validation – nN predictions vs. FE results for multiple design/impact variations

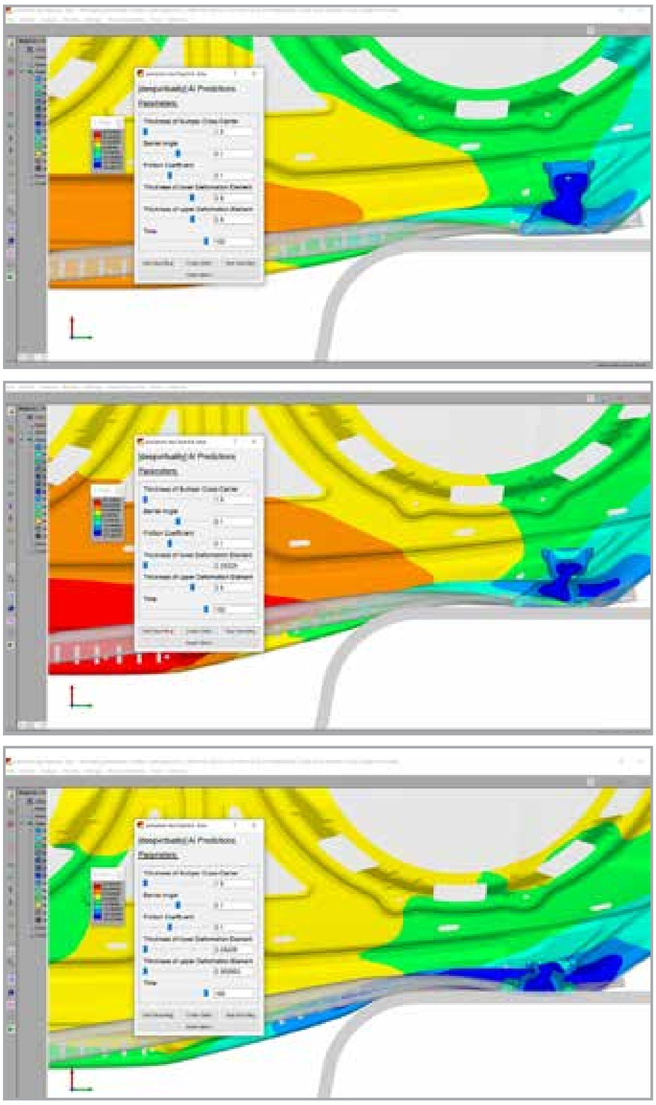


Fig. 8 - e-fea GUI featuring slider-based design variations with live nN predictions e.g. thickness variations of the upper and lower crashbox elements

Conclusion

Sophisticatedly trained on existing simulation and/or testing data, nNs are able to accurately predict the mechanical properties of all products – from stiffness and durability, to acoustics, fluid dynamics, crash behavior and more.

Tailored nN solutions featuring instant design variations with live AI predictions (see Fig. 8) enable massive reductions in product development times and costs, as well as significant improvements in product quality by leveraging the value of simulation techniques and even replacing simulation in many areas!

Consequently, AI and nN are about to disrupt today's design and engineering processes and trigger a new era in the multiple fields of product development, across all industries.

For more information, visit:
www.payerconsulting.com

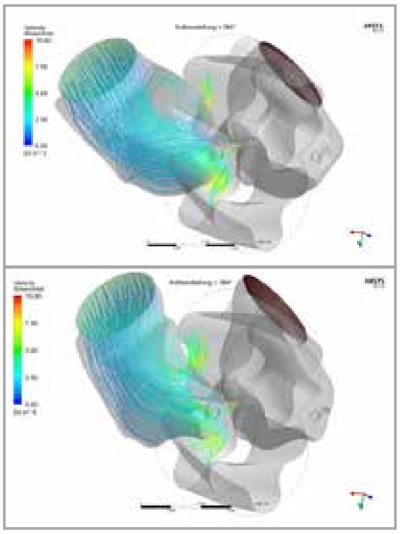
CFX Berlin's TwinMesh enables standardized design for new rotary lobe pumps

CFD analysis improves performance, durability and service life of pumps for Vogelsang GmbH & Co. KG

By Steffen Knabe
Vogelsang GmbH & Co. KG

Since the early 1970s, when rotary lobe pumps were invented mainly for agricultural purposes, their application has expanded from fertilizer distribution in agriculture to disposal and wastewater management in civil engineering and general industry. Rotary lobe pumps are the most reliable pumps in agriculture but, like all other technical equipment, they have to be optimized and constantly re-designed to continue competing in the market.

Traditionally, pump development involves intensive physical testing, but modern engineering offers much more; for example, computational fluid dynamics (CFD). Steffen Knabe, senior engineer at Vogelsang GmbH & Co. KG, one of Germany's leading rotary lobe pump manufacturers, uses TwinMesh for CFD analysis to improve the performance, durability, and service life of their rotary lobe pumps. Thanks to modern engineering, which uses state-of-the-art virtual product development tools, the newly developed IQ152 series weighs less, consumes less power, and has greater efficiency compared to previous models. This article describes the tools used by the engineering team at Vogelsang GmbH to make engineering easier. Finite Element Method (FEM) is a well-established engineering tool in the early development phases of rotary lobe pumps at Vogelsang GmbH. In contrast, CFD has mainly been used, until now, to optimize the inlet and outlet ports of their complex pumps. Due to the excessive man-hours required to ensure high quality CFD results, engineers often have to use reduced models with poor mesh quality to represent their machines. Thanks to the TwinMesh meshing tool from CFX Berlin, mechanical engineers now have a tool that allows them to analyze and optimize all their positive displacement (PD) machines without compromise. Axial and radial gaps, which have a serious influence on the prediction of machine performance, cannot be neglected in CFD analyses. Furthermore, the ability to use structured grids and have reasonable resolution times for simulation are also essential. This is why TwinMesh is a key tool for solid, reliable and accurate simulations and, in Vogelsang GmbH's case, for the development and optimization of rotary lobe pumps. For the newly developed IQ152 series, Vogelsang GmbH performed extensive physical experiments to validate their CFD results. In fact, during those experiments, the influence of the revolution speed on the torque was investigated and was found to be almost perfectly matched by the simulation data. In addition, the experimental data showed that changing the revolution speed also impacted leakage flows, which was also confirmed by post-processing the simulation data. The figure below shows the streamlines in the inlet area of the new IQ152. The simulation data revealed dead zones in that area as well as secondary vortices which resulted in a reduced coefficient of performance (CoP), higher energy consumption, and had a negative influence on cavitation behavior.



As a result, the engineering team was able to identify an inlet design to eliminate these dead zones and the undesirable vortices (see figure). Another advantage of the new design was the reduction in the material of the cast housing, which led to a lower pump weight and lower production costs for the new rotary lobe pump series. In addition, the simple TwinMesh and CFD setup made it possible to easily vary the rotor design to find a design with low cavitation behavior. All in all, the new design extends the service life of the pump, which reduces maintenance time and frequency, to the advantage of Vogelsang GmbH's customers. In summary, we can recommend TwinMesh to all manufacturers of PD machines and have decided to use CFD and TwinMesh for the development of the forthcoming Vogelsang GmbH rotary lobe pumps.

For more information
www.twinmesh.com or www.cfx-berlin.de
and contact Alessandro Arcidiacono - EnginSoft
a.arcidiacono@enginsoft.com

Conduct fatigue checks directly in Ansys Mechanical using SDC Verifier

Template-based design automates future routine work, and automated documentation facilitates report generation

By Oleg Ishchuk
SDC Verifier

A lot of steel structures such as cranes, heavy machinery and other equipment subjected to repetitive loading is highly likely to develop cracks or failures because of the fatigue damage. The cost of every hour of equipment downtime is usually very expensive, making it mandatory to perform residual life calculations or fatigue checks.

Typical crack due to fatigue damage.

The calculation procedure usually consists of the following important steps:

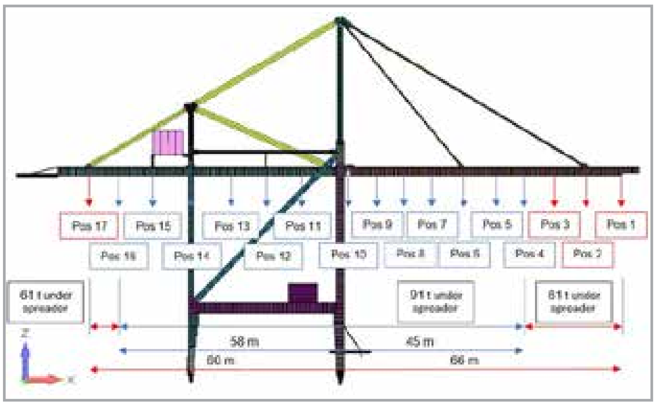
1. ***Gathering information*** about the working conditions, loading history and material data, and then preparing the Basis of Calculation including loads, boundary conditions, load combinations with partial load factors, and safety factors.
2. Recognition of the welds and setting the weld ***classification***, material properties and defining the working conditions.
3. ***Verification*** of fatigue damage according to the standards and rules to guarantee the safety and reliability of the structure.
4. Modifications to the ***design*** to achieve the desired results, optimize the design, or save costs.
5. ***Documentation*** of the results for submission to a certification organization/the customer.

With the help of SDC Verifier, this procedure is completely automated and can be performed directly in the Ansys Mechanical model.

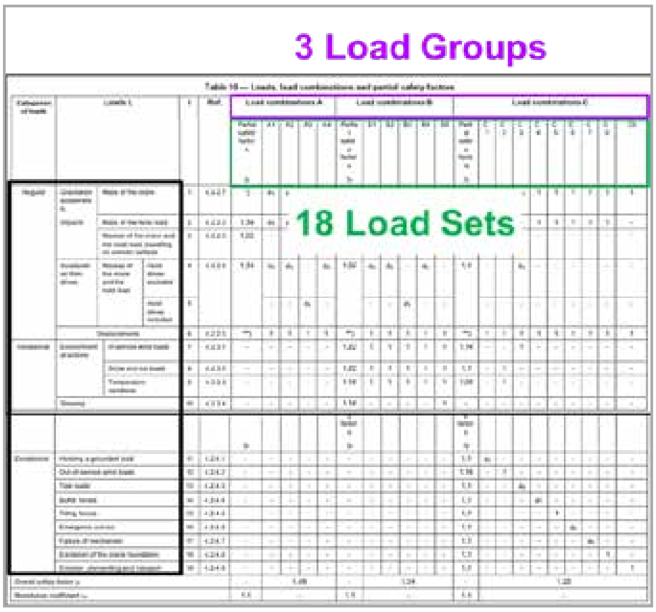
1. Basis of calculation

It makes no difference whether it is a new design check, a residual life analysis or a crack investigation – it is very important to collect and properly transfer to the finite element analysis (FEA) model all necessary information about the working conditions and loading history of the structure (or the estimated loads for a new design). A ***realistic Basis of Calculation*** results in more precise calculation results.

The combination of loads and impacts is usually set by the standards and requires several groups of linear combinations of



Schematic description of the loads on the STS crane



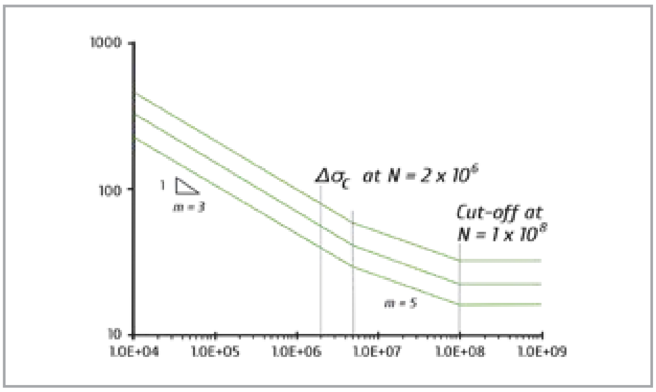
Load combination table from the Standard

single load results. For example, according to FEM 1.001 fatigue testing, an engineer should combine: gravity, effective loading, acceleration and dynamic impact in all possible scenarios, taking into account partial load factors, dynamic and gain coefficients for the Strength group of load combinations with a safety factor of 1.5. All the same conditions plus the effect of wind must be combined for the Wind group of load combinations with a safety factor of 1.33. The next group will be an exceptional load scenario where the same effects have different partial load factors and a safety factor of 1.1.

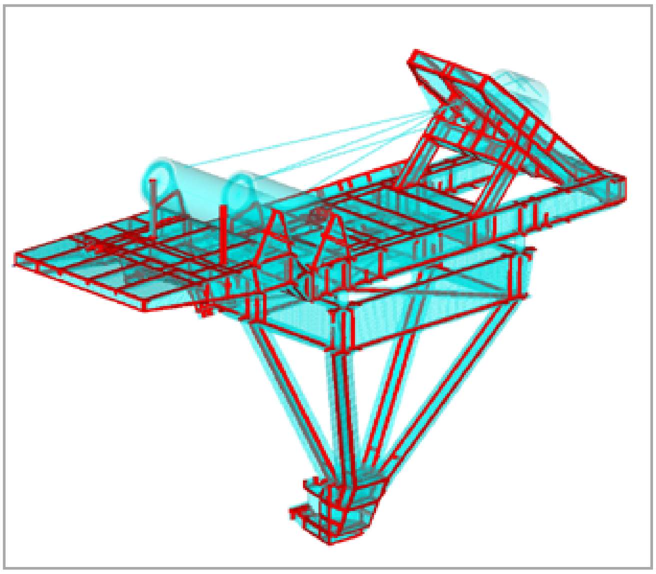
At this point, the SDC Verifier offers the engineer a user-friendly interface for combining the loads into a matrix of load sets and groups, taking into account all the safety factors and partial load factors. Not only does this simplify the processing of a large amount of load history data, but it also shortens the Ansys Mechanical solution time because each load effect will be solved once with Ansys only and then linearly combined using SDC Verifier.

2. Setting the classification

In the second stage, there are three important characteristics that should be set to obtain realistic fatigue verificationresults:



S-N curve graph



Example of SDC Verifier's weld recognition

- weld type or fat class (notch grade) – which determines the quality of the weld;
- material – which determines the grade of steel that the structure is made of; and
- element group – which determines the intensity of load and usage of the verified structure.

Weld type is crucial because fatigue problems are most likely to occur on welds. Therefore, in the S-N curve fatigue analysis method it is necessary to establish a classification based on a weld type to assign a curve. The general curve of the stress level against endurance is shown below.

The first challenge is determining which elements of your FEA model are welds. The SDC Verifier Weld finder tool solves this problem by introducing automatic one-click weld recognition. It automatically finds all welds in the FEA model based on thickness change, material/property change, joint angle and user-defined rules or exceptions.

After recognition is complete, the classification must be determined for all welds in the model. SDC Verifier's selector helps you create components for all welds, weld intersections, weld tips, etc. Moreover, since the Weld finder reorients the stresses and forces in the welding directions, different classifications can be made along the weld, perpendicular to the weld, and for shear.

Element(s) Classification

No.	Selection	Classification
1	Full Model	W0
2	All welds	K1 (X)
3	All welds	K3 (Y)
4	All welds	K0 (XY)
5	All welds intersections	K2 (X)
6	All Welds Tips	K3 (All Directions)
7	Component '6..Stiffeners we...	K2 (X)
8	Component '6..Stiffeners we...	K3 (Y)
9	Component '25..TIPS Midse...	K3 (X)

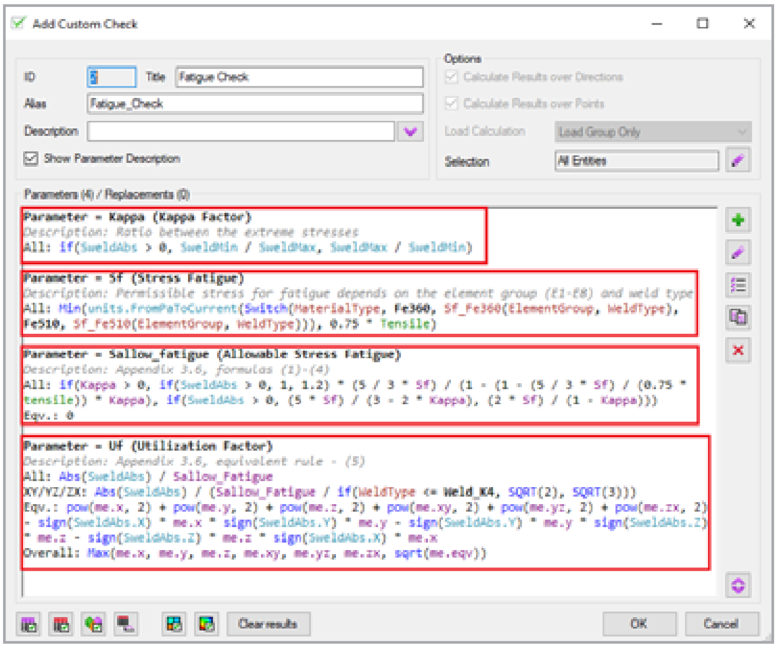
Weld classification example

Material type is often sourced from Ansys Mechanical or can be easily set in SDC Verifier (there is an automatic reminder if some of the material properties are not set).

3. Standards-based verification

The next step is the verification itself. SDC Verifier performs the fatigue check according to different standards in the Offshore, Heavy lifting, Heavy machinery, Maritime, Shipbuilding and Civil industries. The following standards are currently available: Eurocode 3 (EN1993-1-9, 2005)

- EN 13001 (2018)
- FEM 1.001 (3rd, 1998)
- DNV-RP-C203 (2016)
- DIN 15018 (1984)
- FKM Guidelines



Fatigue check formulas editor

SDC Verifier uses the Allowable Stress Design method. All formulas are open, so the user can see all the formulas behind the calculation kernel, including descriptions and explanations of where they have been taken from.

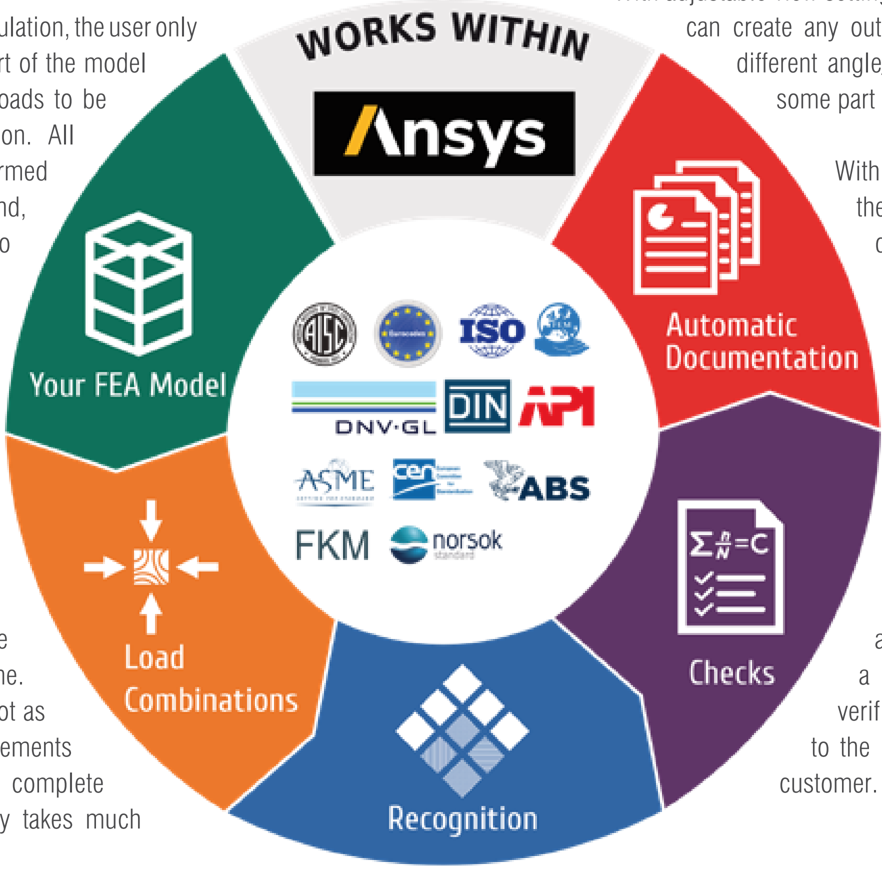
Full control over the structure of the tables makes it possible to present the maximum or detailed output result for the complete model or individual components over the loads and combinations, or to prepare an overview table of the various load parameters. With adjustable view settings and selectors, the user can create any output or check result at a different angle/view for the full model, some part or cut.

Before running the calculation, the user only needs to select the part of the model to be tested and the loads to be used in the calculation. All calculations are performed in the background, however it is possible to access a log file of the calculation details if full control of the analysis is required.

With SDC Verifier that allows the code to be verified directly in Ansys, there is no longer a need for routine manual calculations, spreadsheets, or complicated testing and verification procedures to check the fatigue damage or to predict the residual life of a structure. In addition, you can generate a full calculation and verification report to submit to the certifying organization or customer.

4. Modifications to the design

In the initial phase, modifications to the design seem to be a minor part of the project execution time. But if the results are not as expected and improvements are required, then the complete review process usually takes much longer than expected.



SDC Verifier Workflow scheme.

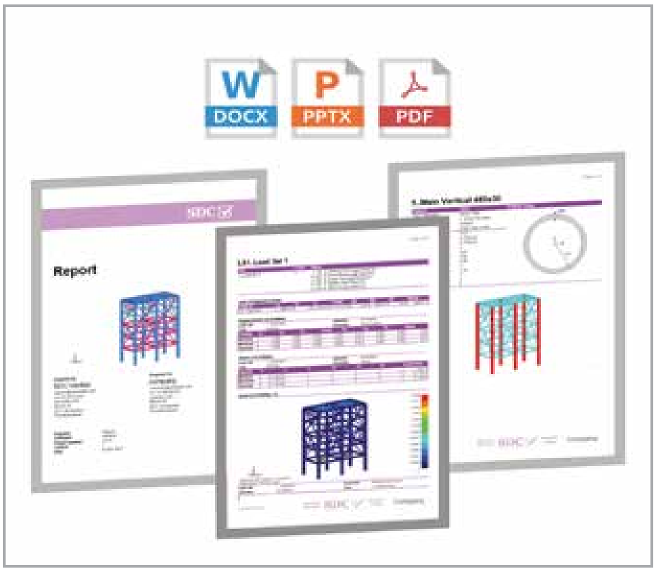
SDC Verifier is template-based, which means that if the check has been done once, then in the future, routine work will be automated on the basis of calculation, recognition, and the calculations themselves. All the user has to do is change the design and solve the model. Subsequently, the data is updated and a report is generated with two mouse clicks, to complete the entire procedure.

This is a huge time saver and reduces deadline pressure because, as we all know, most design modifications and improvements are done at the last minute!

5. Documentation automation

Last, but not least, is the automatic reporting. Few people like writing reports. However, there is still a great deal of documentation that must be provided to a manager, customer or certification organization.

The SDC Verifier Report designer helps you to create a template-based document that contains the following info: model description, loads and boundary conditions, Ansys results output, and code validation results. All data can be presented with an extensive range of plots and tables.



The automatic report can be exported to Word, PowerPoint or PDF

For more information:
Oleg Ishchuk - SDC Verifier
oleg@sdcverifier.com or visit:
www.sdcverifier.com

Investigating wind-turbine structural behavior under icing conditions

Multiphase fluid-structure interaction for 3D wind-turbine blades

By Luis Rubio and Laszlo E. Kollár
Savaria Institute of Technology

This technical article presents a comprehensive methodology for studying 3D wind turbine blades to investigate their dynamic behavior under icing conditions. The governing equations of the system which correspond to a multi-phase fluid composed of cold winds and supercooled droplets were solved considering the conservation of mass and conservation of momentum of the multi-phase flow in a predetermined domain. The solution to these equations provided the pressure distributions over the blades, which were then used as a load to calculate the mechanical behavior of the iced blades.

The blades are composed of an orthotropic composite material used lately by the Eolic industry. The angular velocity of the blade was considered as a centripetal load. During the structural analysis of the blades, the deflection/deformation of the blade, the equivalent stress (Von-Misses) and the force and moment reactions during rotation were studied.

The planet's environmental problems, energy access and energy security issues, the geo-politics of oil and the resulting conflicts in many parts of the world all point to the urgent need for a transition from the predominant hydrocarbon-based energy system to a renewable energy-based system in which wind energy plays the predominant mainstream role. Numerical methods are increasingly used to study complex systems to save time and money in industry today.

This article presents a comprehensive numerical methodology to study 3D wind turbine blades and to investigate the system's structural behavior under icing conditions. The clouds responsible for such conditions are simulated with multi-phase flow, to ensure a comprehensive approach.

The commercial finite element software Ansys, including the Fluent and Mechanical packages, was used for this purpose. Computational fluid dynamics (CFD) was used to solve the airflow equations, i.e. the conservation of mass and conservation of momentum, over a predetermined domain.

The SST k-omega turbulence model was used and the wind velocity was assumed to be constant. The model calculated the pressure distribution on the surface of the wind turbine blade. This was imported as a load into the finite element analysis (FEA) to test the mechanical behavior of the blades. The angular velocity of the blade was considered as a centripetal load in this analysis.

The blades were composed of an orthotropic composite material recently in use in the Eolic industry. The FE analysis revealed the deflection/deformation of the blade, equivalent stress (Von-Misses), and the force and moment reactions. The numerical results generated in Ansys are presented and analyzed and are compared to approximate manual calculations, based on the data obtained in the simulations, in order to evaluate the accuracy of the simulation results.

Methodology

This study aimed to test the mechanical behavior of the blades of the wind turbine under two loads: the pressure distribution on the blades resulting from the effect of the air, and the centripetal force resulting from the angular velocity of the wind turbine blades. References [1] and [2] explain this procedure in more detail.

Step 1, Aerodynamics (CFD): The aerodynamic load on the blade was considered. The air flow was defined in a suitable computational domain. This domain was meshed and a constant free stream velocity was prescribed at the inlet to the computational domain. The governing equations were continuity (conservation of mass) and Navier-Stokes (conservation of momentum) including the Coriolis and centripetal forces. The frame of reference was set to rotate with the blade to avoid having to create a moving mesh in the model setup. Furthermore, the Reynolds Averaged form of continuity and momentum was used with the SST k-omega turbulence model to close the equation set.

Step 2, Blades (FEA): The pressure values applied to the wetted areas of the blade were introduced as pressure loads for the mechanical analysis of the blade, and the stresses and deformations of the blade were analyzed. The blade geometry

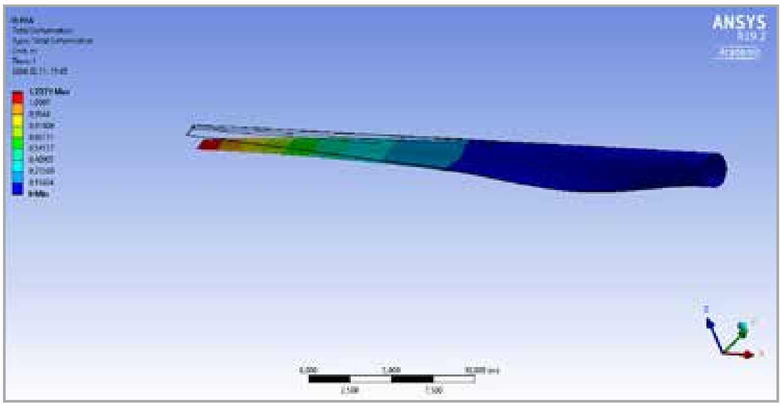


Fig. 1a - Tool deflection caused by wind air (case I).

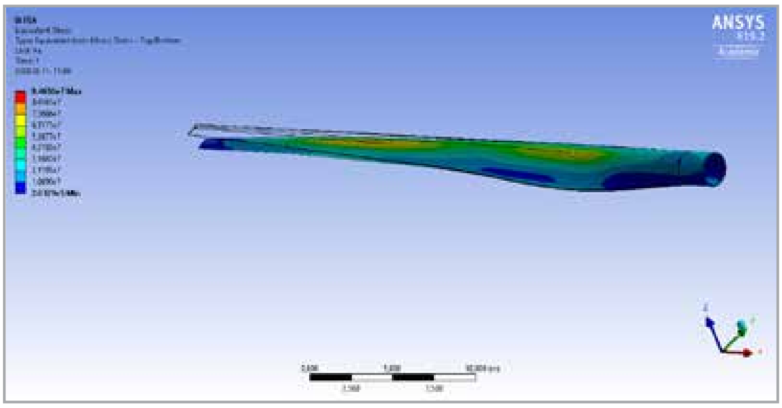


Fig. 1b - Tool stress caused by wind air (case I).

was defined, it was meshed and the pressure load was applied. The shell theory for curved surfaces was used to calculate the deformation and the stresses on the blades. Moreover, the blade's connection to a hub was also simulated to extract information about the forces and bending moments.

As mentioned above, the effect of blade rotation was taken into account using centripetal loading. Large deformation increases blade stiffness; thus, the faster the blade rotates, the more rigid its material behavior becomes.

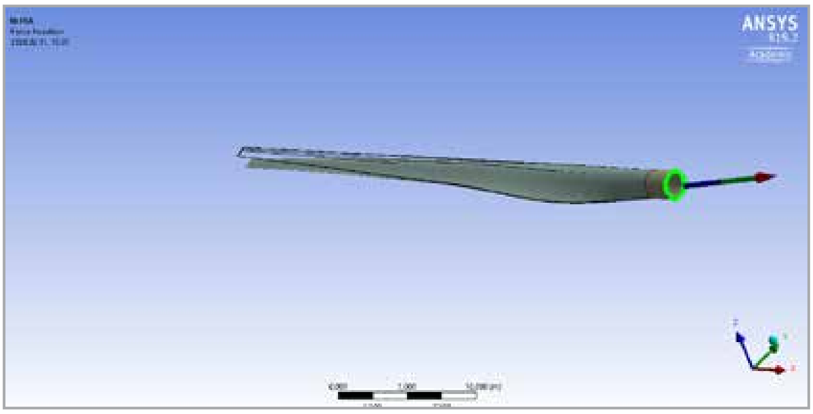


Fig. 2a - Forces on support of the blade (case I)
Fx = 1,448e+006 N ; Fy = 8914,5 N ; Fz = 1,9662e+005 N ; F = 1,4613e+006 N

Results

Deformations/deflections and the equivalent (Von Misses) stresses of the blade were obtained in static equilibrium. Furthermore, the force and moment reactions in the connection between the blade and the hub were also calculated. These results contribute to the understanding of what is happening in the system under the conditions considered.

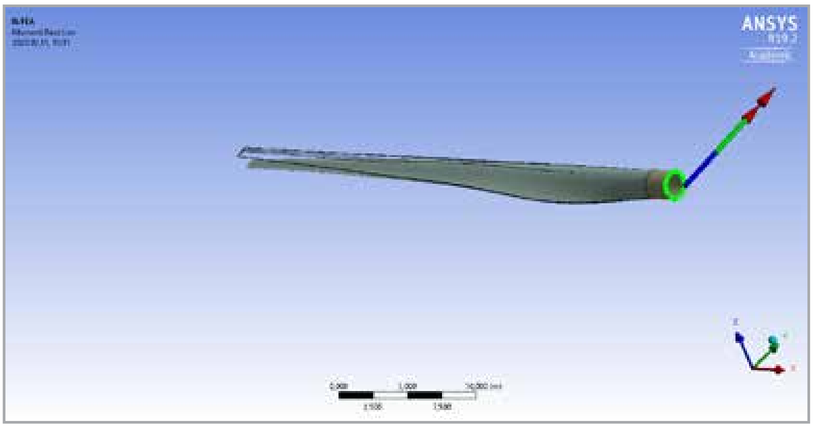


Fig. 2b - Moments on support of the blade (case I)
Mx = 52133 N·m ; My = 5,925e+006 N·m; Mz = 4,0336e+005 N·m ; M = 5,9389e+006 N·m

The coordinate system was defined so that the span of the blade is in the direction of the X-axis, and the blade rotates around the Z-axis (see Fig. 1a and Fig. 1b). As expected, the X-component of the force was the largest, whereas the moment's largest was the Y-component. Fig. 1a and 1b illustrate the deformation and the stress on the blade. Fig. 2a and 2b show the force and moment components.

Multi-fluid analysis

Methodology

Various CFD approaches are used to couple aerodynamics and icing to assess the stability and control of a wind turbine blade under icing conditions or to generate a CFD database of wind turbine icing simulations. For this study, we used the FENSAP-ICE approach, which views ice accretion simulation as a combination of the following methods:

- Calculating the flow around the bare blade with a 3D incompressible flow, which offers an acceptable approximation for velocities in the range of interest (i.e. up to about 30 m/s), by applying turbulent Navier-Stokes equations (using any CFD code; here with Ansys Fluent and FENSAP).
- Calculating the collection efficiency of the water droplets on the blade surface using the 3D Eulerian method (here with DROP3D).
- Solving 3D heat and mass transfer over the blade surface using partial differential equations to predict 3D ice accretion patterns (here using ICE3D).

All three of these approaches are based on partial differential equations (PDE) [3].

In addition, the displacement grid provides information on the new mass distribution of frozen droplets over the blade. This information can be used to redistribute additional layers of ice over time, or to analyze the new structural form of the iced blade.

Results

The FENSAP ICE package outputs show the droplets and distribution of the ice mass over the 3D wind turbine blade for the two cases in Table 1 (See Fig. 3 (a-d)).

Further research

Further research aims to analyze the structure of the blades to determine the deflection/deformation of the blade, the equivalent stress (Von-Misses), and the force and moment reactions under both normal and icing conditions. Additional simulations will

Icing condition	Wind speed (m/s)	Air Ta (°C)	Liquid Water Content (gr/m^3)	Median Volume Diameter (μm)
In-cloud icing	20	-10	0.3	27
Freezing drizzle	10	-5	1.5	62

Table 1 - Icing conditions [4], case I: in-cloud icing, case II: freezing drizzle

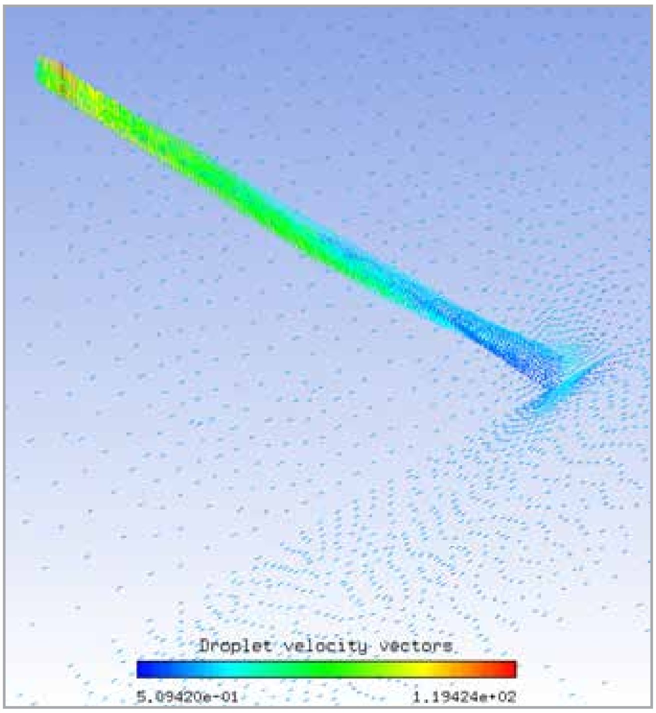


Fig. 3a - Droplet velocity vectors around and on blade (case I).

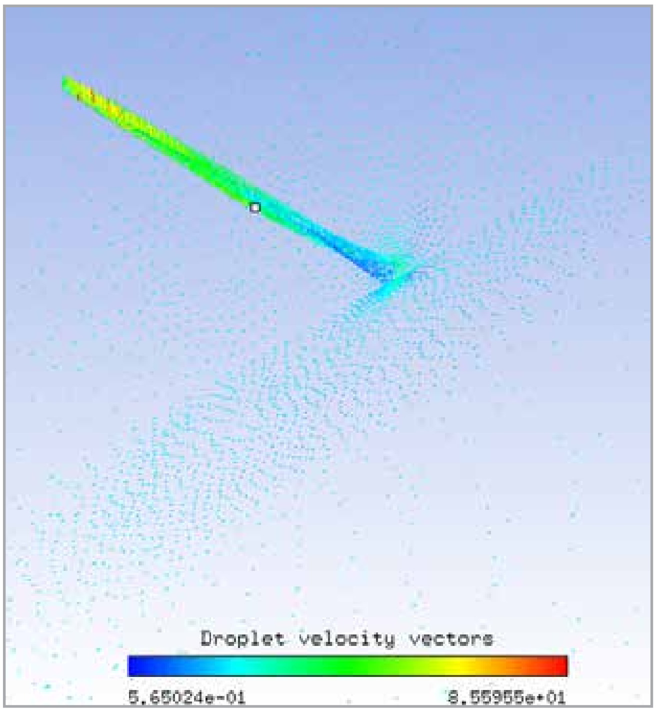


Fig. 3c - Droplet velocity vectors within the domain (case II).

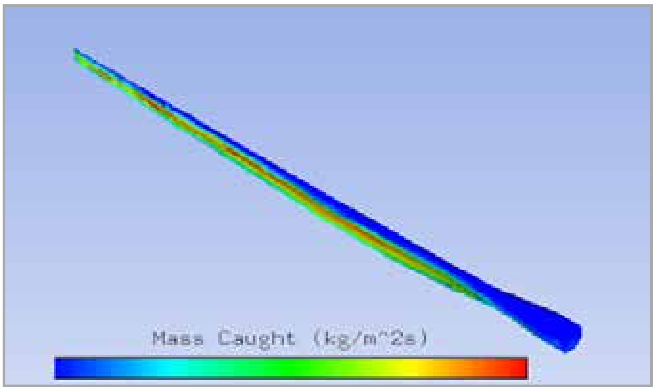


Fig. 3b - Ice mass caught on blade (case I).

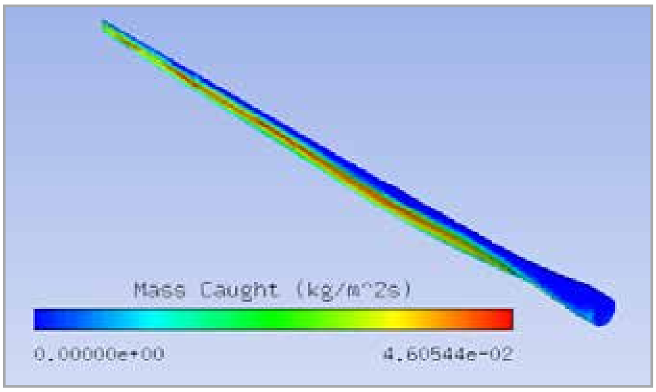


Fig. 3d - Ice mass caught on blade (case II).

be performed with Ansys/FENSAP for this purpose, taking into account the distribution of the ice mass over the blade using the displaced grid obtained.

The steps include displacing the grid, calculating the aerodynamic solution that best accounts for the effect of accreted ice and exporting the pressure distribution for the mechanical analysis of the blade.

Conclusions

This article describes the mechanical analysis of wind turbine blades under different atmospheric conditions. Normal conditions were considered using single-phase flow, whereas icing conditions were simulated using multi-phase flow.

The finite element software Ansys appeared to be a very efficient tool for such simulations. The results of the fluid-structure interaction analysis and the multi-fluid analysis were presented. Further studies to obtain the mechanical behavior of the iced blades are under consideration.

Acknowledgments

The research described in this article was executed under the terms of the project "EFOP-3.6.1-16-2016-00018 – Improving the role of research + development + innovation in higher education through institutional developments assisting intelligent specialization in Sopron and Szombathely".

References

[1] Bhaskaran, R., A hands on introduction to engineering simulations, Cornell University and Ansys, 2016, (https://www.edx.org/es/course/a-hands-on-introduction-to-engineering-simulations).

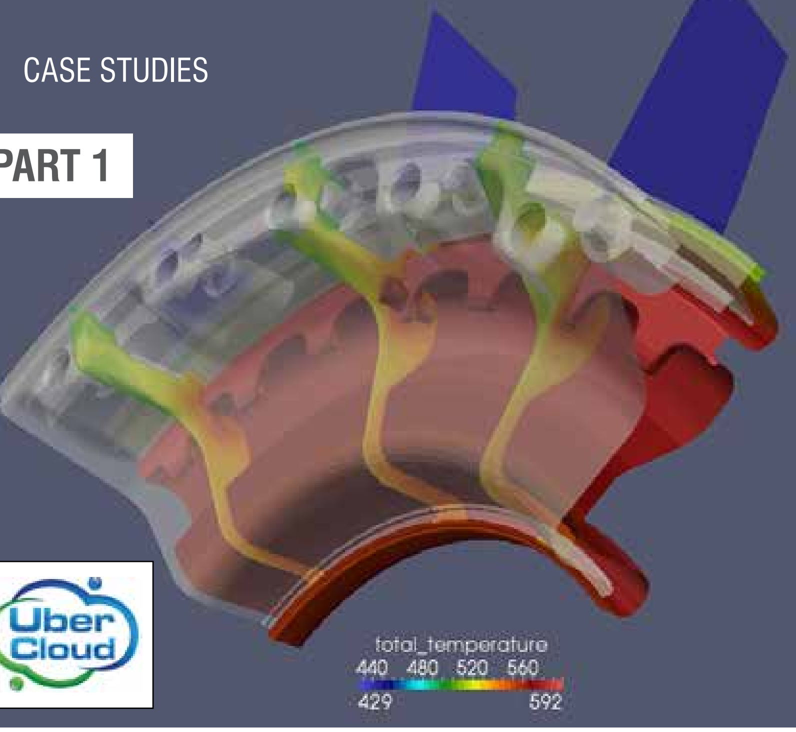
[2] Ansys Workbench 19.2, Workbench User's Guide (https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v192/wb2_help/wb2_help.html).

[3] Ansys-Fensap Tutorial Guide 19.2 (https://www.ansys.com/products/fluids/ansys-fensap-ice).

[4] Kollár, L.E. and Mishra, R., Inverse design of wind turbine sections for operating under icing conditions, Energy Conversion and Management, Vol. 180, pp. 844-858, 2019.

For more information
lrr@inf.elte.hu or kl@inf.elte.hu

PART 1



Aerodynamics Simulations with Ansys CFX and Fluent in the UberCloud

By Wolfgang Gentzsch
UberCloud

While cloud-based services have been growing in popularity since the launch of Amazon's cloud resources in 2006, their uptake in the engineering sector has been much slower. Since then, many of the barriers to entry for the engineering community, such as lack of security, traditional software licensing models, and internal resistance from conventional management structures, have been removed. In this article, UberCloud presents two aerodynamics case studies dealing with cloud-based services for engineering-specific applications and use cases that objectively demonstrate the progress of cloud computing in the aerodynamics sector over the past few years.

Engineering companies only recently became aware of the promise of the cloud computing model. Granted, the major reasons for this hesitation were a few severe roadblocks that were identified in the past such as security concerns, traditional software licensing models (e.g. annual, perpetual), large data transfers, the engineer's concern of losing control over their simulation assets (e.g. software, data), and internal resistance from management and IT departments who often barricaded themselves behind old compliance regulations not yet adapted to modern times.

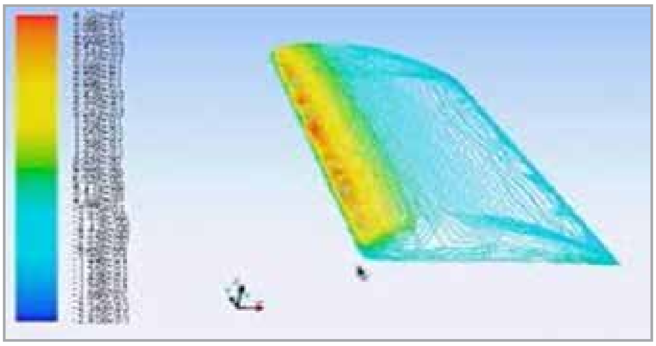
In 2012, UberCloud started a series of cloud projects in engineering simulation, moving complex simulation workflows to the cloud with the goal of analyzing the benefits and challenges, and publishing the lessons learned with recommendations for the engineering community [1].

These projects included defining the engineering application use case, moving the application software to the cloud, running the simulation jobs in batch or in interactive mode, evaluating the results via remote visualization, transferring the final results back onto premises, and writing the case study.

Since then, we have performed 220 high performance computing (HPC) cloud projects and published 125 case studies, 25 of them based on Ansys software such as CFX, Fluent, Mechanical, LS-DYNA, HFSS, and Discovery Live, as demonstrated in our previous two articles in the EnginSoft Newsletter, [2] and [3].

The Ansys and other case studies are available for download [4]. We have also been able to remove almost all the roadblocks by developing a software technology that hosts the engineer's complex workflows in an isolated software container that sits on a dedicated and secure HPC resource in any cloud – public, private, hosted, and hybrid.

This article presents two aerodynamics case studies based on Ansys CFX and Fluent in the cloud, by briefly introducing the engineering use case, describing the cloud implementation, benefits and challenges, and concluding with the lessons learned and recommendations. The project team for these two projects



"I've been using cloud computing for several years, tried four different cloud providers and found this UberCloud service for CFX and Fluent intuitive and very easy to use. I didn't expect it would be so easy."

consisted of computational fluid dynamics (CFD) expert Praveen Bhat, Technology Consultant in India; software provider Ansys with trial licenses for the CFD codes CFX and Fluent; cloud resource providers Advania Data Centers for the first use case and ProfitBricks for the second case; and technology experts Reha Senturk, Ender Guler, and Ronald Zilkovski from the UberCloud.

Aerodynamic study of a 3D wing using Ansys CFX

USE CASE

The aerodynamic study on the aircraft wing investigated the air flow and the forces acting on the wing due to air velocity to optimize the wing's profile and quantify the lift and the drag forces. The CFD analysis provided in-depth insights on the air flow, pressure and velocity distribution around the wing and the parameters required to calculate the lift and the drag forces. A standard wing profile was considered for this project. The cloud environment was accessed using a VNC viewer through the engineer's web browser. Ansys software was running in UberCloud's application software container.

The following flow chart defines the container setup and the modelling approach for setting up and running the simulation in the Ansys containerized environment:

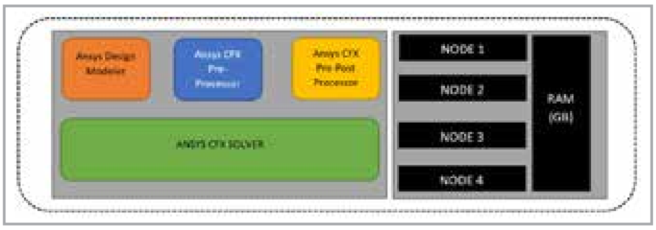


Fig. 1 - Container environment with Ansys CFX application

The model construction and setup is shown in the following flow chart:

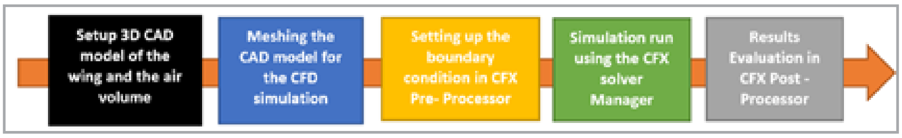


Fig. 2 - Stages in model setup and simulation run in Ansys CFX

Fig. 2 describes the step-by-step approach to set up the CFD model using the Ansys Workbench environment. The Ansys CFX simulation was setup in the Advania HPCFLOW cloud environment. The following snapshot highlights the wing geometry considered and the CFX mesh model.

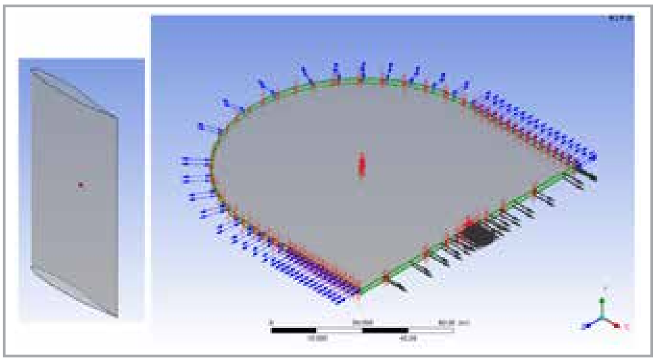


Fig. 3 - 3D geometry of the wing

Model Size	No of Nodes	No of cores for each node	No of cores	Solution time (sec)
10 million elements	1	32	32	246.4
	2	32	64	134.3
	3	32	96	100.8
	4	32	128	85.03

Table 1 - Simulation time (in secs) for 10 million elements and different no. of cores

Model Size	No of Nodes	No of cores for each node	No of cores	Solution time (sec)
100 million elements	1	32	32	4341
	2	32	64	2321
	3	32	96	1258
	4	32	128	906

Table 2 - Simulation time (in secs)) for 100 million elements and different no. of cores

HPC Performance Benchmark

The aerodynamic study on the aircraft wing was carried out in an HPC environment built on a 128-core server with 250 GB RAM, CentOS Operating System, and Ansys Workbench 19.0. The server performance was evaluated for different numbers of discrete elements. The run time can be reduced by using a higher number of cores.

Table 1 and 2 highlights the solution time for a 128-core system with up to 100-million elements.

Aerodynamics and fluttering of an aircraft wing with fluid-structure interaction

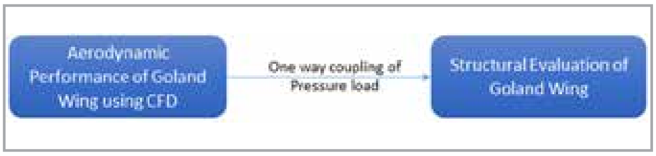
USE CASE

Fluid-structure interaction problems are highly complex and require modelling of both fluid and structure physics. In this case study, the aeroelastic behavior and flutter instability of an aircraft wing in the subsonic

"The whole user experience in the cloud was similar to accessing a website through the browser."

incompressible flight regime were investigated. The project involved evaluating the wing's aerodynamic performance using CFD. The standard Goland wing was considered for this project. The simulation platform was built in a 62-core HPC cloud with the Ansys modelling environment. The cloud environment was accessed using a VNC viewer through a web browser.

The following flow chart defines the fluid structure interaction framework used to predict the wing performance under aerodynamic loads:



The Fluent simulation setup was solved in the HPC Cloud environment. The simulation model needed to be precisely defined using a large amount of fine mesh elements around the wing geometry.

The pressure load calculated from the CFD simulation was extracted and mapped onto the Goland wing to evaluate the structural integrity of the wing.
The following steps define the procedure for the structural simulation setup in Ansys Mechanical:

1. The Goland wing was meshed with Ansys Mesh Modeler and hexahedral mesh models.

2. The generated mesh was imported into Ansys Mechanical where the material properties, boundary conditions etc. were set up.

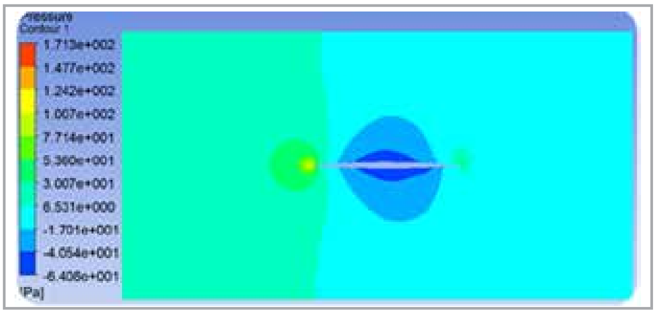


Fig. 4 - Pressure distribution at mid-section of wing

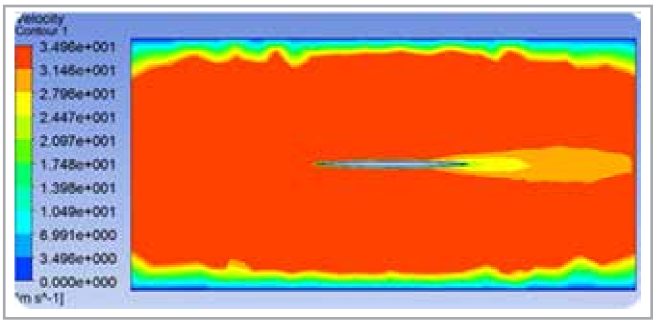


Fig. 5 - Velocity distribution at mid-section of wing

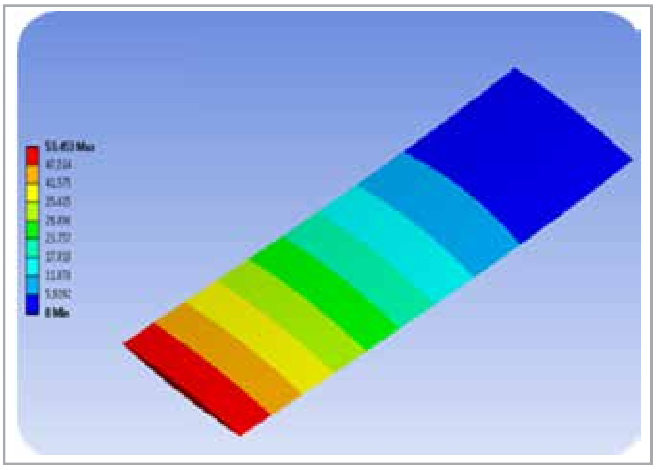


Fig. 6 - Aerodynamic loads acting on wing wall

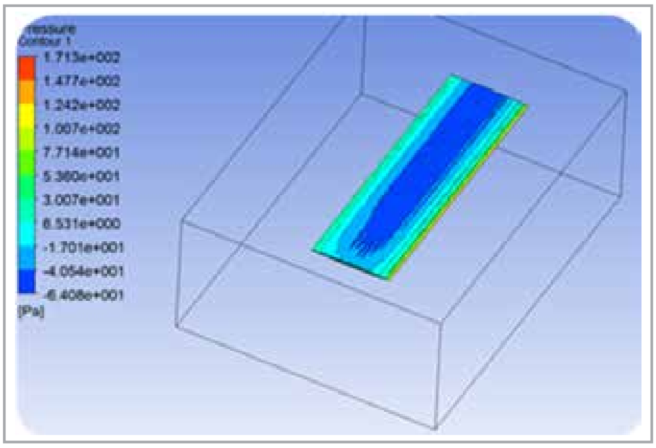


Fig. 7 - Wing deflection due to aerodynamic load

3. The solution methods and solver setups were defined. The analysis setup mainly involves defining the type of simulation (steady state in this case), output result type (stress and displacement plots, strain plots etc.).

4. The pressure load extracted from the CFD simulation was mapped onto the wing structure to evaluate the wing behavior under aerodynamic loads.

Fig. 4 shows the pressure distribution at the mid-section of the Goland wing. The pressure distribution across the section is uniform.

The velocity plot in Fig. 5 shows that the air velocity varies near the edge of the wing. The air particle velocity is uniform with particles following a streamlined path near the wing wall.

Figs. 6 and 7 indicate the aerodynamic loads on the wing, which are calculated based on the pressure distribution on the wing wall. The respective aerodynamic loads were mapped onto the wing structure and the deformation of the wing was simulated to evaluate the wing deformation.
The wing's behavior under the aerodynamic loads indicates its flutter stability.

HPC performance benchmarking

The flutter stability of the aircraft wing was studied on a 62-core server with CentOS Operating System and Ansys Workbench.

The server performance was evaluated by submitting the simulation runs for different numbers of elements. The run time can be reduced by using higher core systems. The following table highlights the solution time captured on 8, 16, and 32 cores with element numbers ranging between 750K to 12 million:

Challenges, benefits, conclusions, and recommendations

CHALLENGES

The challenge was scaling the engineering workstation environment to a multi-node multi-container HPC server environment. Another challenge in these two projects was the accurate prediction of the wing behavior under the aerodynamic forces, which was achieved by identifying an appropriate element size for the mesh model. The finer the mesh, the higher the simulation time, unfortunately, and hence the challenge was to perform the simulation within a reasonable time.

BENEFITS

1. The integrated HPC cloud environment with Ansys 19.0 Workbench made the process of model generation much easier, with process time reduced drastically because of the more powerful HPC resource available in the Cloud.

3. The computation requirement is large for a very fine mesh (e.g. 100 million cells), which is nearly impossible to achieve on a normal workstation. The HPC cloud made it feasible to solve very fine mesh models with drastically reduced simulation times, thereby providing the advantage of obtaining the simulation results within acceptable run times (in the order of three hours).

4. Using Ansys Workbench helped with performing different iterations by varying the simulation models. This further helped to increase the productivity of the simulation setup and thereby provided a single platform to perform end-to-end simulation model development.

5. The benchmarks demonstrated the immense potential of cloud computing, providing greater confidence to setup and run simulations remotely in the cloud. The different simulation setup tools required were installed in the containerized HPC environment and this enabled the user to access the simulation tools without any prior installations.
6. The use of VNC Controls in the Web browser made it very easy to access the HPC cloud without the need for installing any pre-requisite software. The user experience was similar to accessing a website through the browser.

7. The UberCloud containers helped with the smooth set up and execution of the project, with easy access to the server resources and enabled continuous monitoring of the job in progress in the Ansys container.

References

[1] Burak Yenier and Wolfgang Gentzsch: Exploring Computational Biology as a Service, http://www.bio-itworld.com/2013/3/12/exploring-computational-biology-service-ubercloud-round-three.html

[2] Wolfgang Gentzsch: Engineering Simulations Using Ansys In The Cloud – Part 1. Summer 2019, pp. 35 – 39, https://www.enginsoft.com/assets/pdf/newsletter/newsletter2019_02.pdf

[3] Wolfgang Gentzsch: Engineering Simulations Using Ansys In The Cloud – Part 2. Autumn 2019, pp. 40 – 43, https://www.enginsoft.com/assets/pdf/newsletter/newsletter2019_3.pdf

[4] DOWNLOAD Ansys UberCloud Case Studies, https://www.theubercloud.com/ubercloud-compendiums

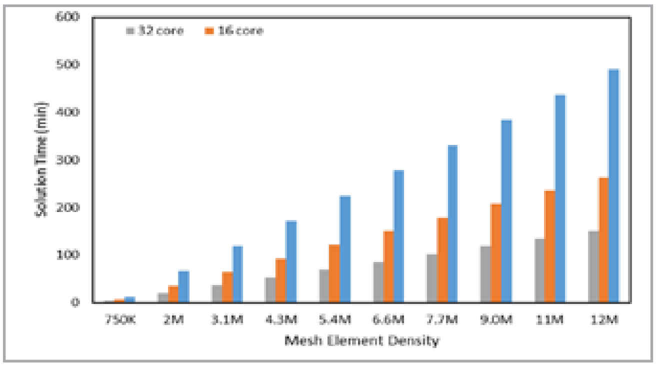


Fig. 8 - Comparison of solution time (mins) for different element density on 8, 16, and 32 cores

CONCLUSIONS AND RECOMMENDATIONS

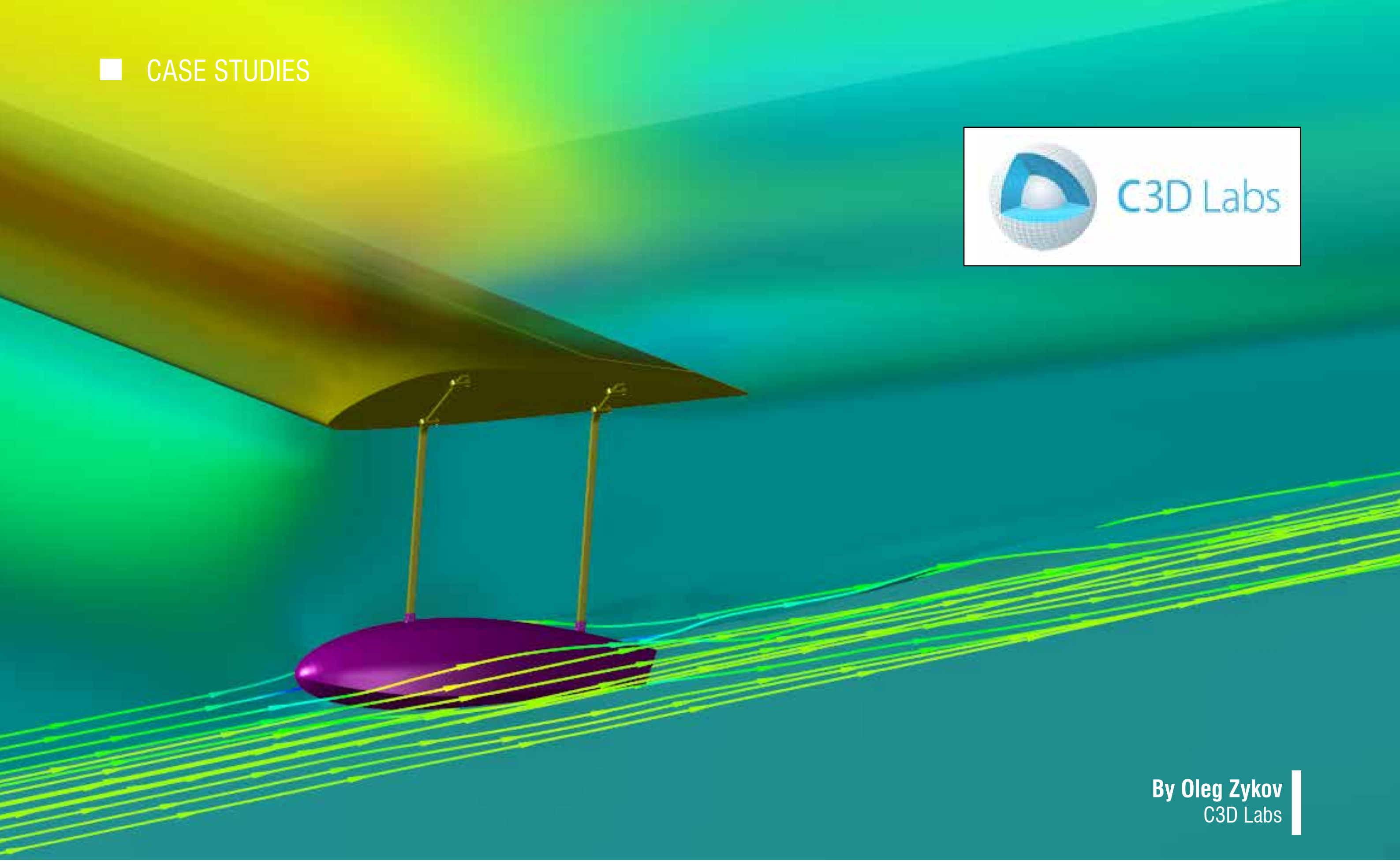
1. The HPC cloud is an excellent fit for performing advanced computational projects that involve technical challenges and require highly scalable hardware resources.

2. Different high-end software tools can be used to perform aerodynamics CFD simulations. Ansys Workbench helped us to solve this problem with minimal effort being required to set up the model and perform the simulations.

3. The combination of the HPC cloud based on HPE and Intel technology, Ansys Workbench, and UberCloud's Ansys container used to host the engineer's complex workflows helped to speed up the simulation set up and execution and also completed the project within a reasonable time.

4. For engineers that want to occasionally burst large simulations to the cloud, or for engineers that may wish to replace an old HPC system (successively) by moving entire HPC projects to the cloud, we invite you to join us for a "Discovery Call", a first proof of concept, a joint cloud architecture design of your customized on-premise/cloud solution, and its final implementation and rollout to production.

For more information
Wolfgang Gentzsch - UberCloud
Wolfgang.Gentzsch@TheUberCloud.com



By Oleg Zykov
C3D Labs

Geometric modeling for CAE developers

Determining the best way to prepare geometric models for analysis and simulation

Developers of computer-aided engineering (CAE) software solves many problems, most of which are associated with a computation kernel. The kernel is responsible for a product's usefulness to customers, ensuring quality, accuracy, speed, and reliability of calculations.

These benefits are not the only essential criteria in a competitive CAE solution. The software must also be capable of:

- Importing CAD models from a variety of sources
- Preparing models for analysis
- Enabling visualization of the entire process and the computation results
- Presenting an engaging application interface

The computation kernel is the key competency and the key differentiator between CAE products, but all developers tend to develop on their own, which results in there being options for other tasks.

The task of preparing a geometric model is accomplished in a variety of ways. Many CAE developers write all the necessary functions on their own. The advantage of this lies in the fact that the software companies therefore completely control the product code and implement the algorithms to do exactly what the developers intend them to. The drawback here is that the in-house approach is time- and resource-intensive and requires programmers highly skilled in the field of geometric modeling. Nevertheless, nearly 100% of new CAE developers adopt this approach and most will not abandon it.

One alternative is to integrate your product with a CAD system via its API. This may well be the easiest approach to reduce the requirement for in-house resources. However, this choice holds you hostage to someone else's set of CAD functions limits your influence over the developers. (The exception is when the CAE vendor buys the entire CAD company, such as when Ansys bought SpaceClaim.) Customers have to install an expensive CAD solution on each user's workstation, even if they only need a small set of the CAD program's functions to prepare the model for CAE.

The other alternative is to license a CAD kernel for your pre-processor. This provides access to a rich set of ready-to-use functions without sacrificing the freedom to make your own development plans. You continue to decide what to implement in your products and how. Kernel licensing enables you to save on resources and reduce programmer requirements. While some of these kernels are expensive, there are many competitive offers on the market. Kernels are available from long-time suppliers Siemens Digital Industries Software (Parasolid) and Dassault Systems Spatial (ACIS and CGM), as well as from modern challengers like C3D Labs (C3D Toolkit) and Kubotek3D (Kosmos Framework). CAE companies ought to carefully examine the capabilities and prices of each of these kernels.

To import geometric models from exchange files, libraries can be licensed from Techsoft (HOOPS Exchange), Datakit (CrossManager), and C3D Labs (C3D Converter). The 3D model's accuracy after it is imported affects the quality of the finite element (FE) mesh that is generated for engineering analysis and calculations. A proven and reliable converter provides tight integration with CAD systems, which is a prerequisite for modern CAE applications.

The display quality of the calculation results affects the perceived user-friendliness of your CAE solution. An external visualization module, such as those from Redway 3D (RedSDK), Techsoft (HOOPS Visualize), or C3D Labs (C3D Vision), provides developers with a number of integrated solutions for maximizing productivity. Examples include functions for controlling the precision of triangulation grid calculations and the levels of image detail.

Why use the C3D Toolkit?

Our C3D Toolkit provides CAE system developers with a set of tools for preparing geometric models for analysis. For instance, C3D Modeler implements all the necessary operations for constructing 2D and 3D objects of any shape. The most sought-after features among CAE developers include Boolean operations, shell sections, split lines, equidistant shell construction, and direct modeling. Key benefits of C3D Modeler are the flexible methods for calculating geometries, such as for constructing triangles, calculating mass-inertial properties or collisions detection between model elements.

The tight integration between our C3D Modeler and C3D Converter modules means that you can correctly import models and related data from standard exchange formats, such as STEP, IGES, ACIS SAT, Parasolid X_T and X_B, STL, VRML, and JT -- and then also export the models and data to other systems. This ensures a high level of interoperability between your CAE application and other third-party software.

C3D Vision provides new capabilities for managing 3D scenes as well as a ready-to-use tree for 3D models and animation, plus interactive tools for scene manipulation in the "user-computer" mode and virtual devices that are integral to modern engineering software interfaces.

Yuri Kozulin, head of C3D Modeler development, states, *"Similar to the developers of other CAD kernels, we originally targeted the functions CAD developers required for the fields of engineering, construction and architecture. However, after enrolling our first CAE customers, we implemented other functions based on their needs. As a result, our C3D Modeler geometry kernel now features tools specifically requested by our CAE industry customers, including:*

- *Storage of double-type and float-type triangulation data*
- *Ability to input triangle parameters by angle, deflection, or maximum allowable distance between two adjacent points*
- *New code samples to demonstrate the rapid creation of split lines without any additional construction*
- *Improved direct modeling for removing fillets, chamfers, and holes.*
- *Control of merging the edges and faces in Boolean operations, cutting shells, and truncating shells.*

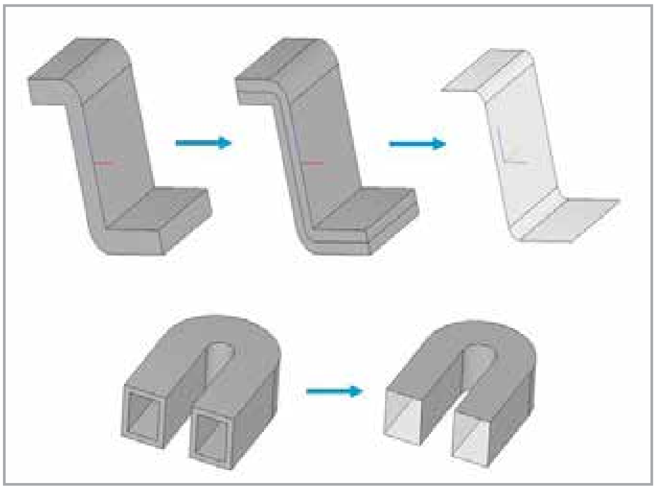


Fig. 1 - Constructing the median shell from given pairs of body faces in C3D Modeler

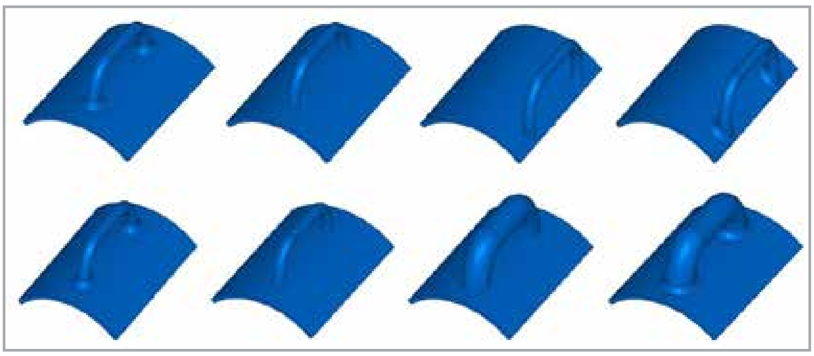


Fig. 2 - Direct geometric modeling in the C3D Modeler

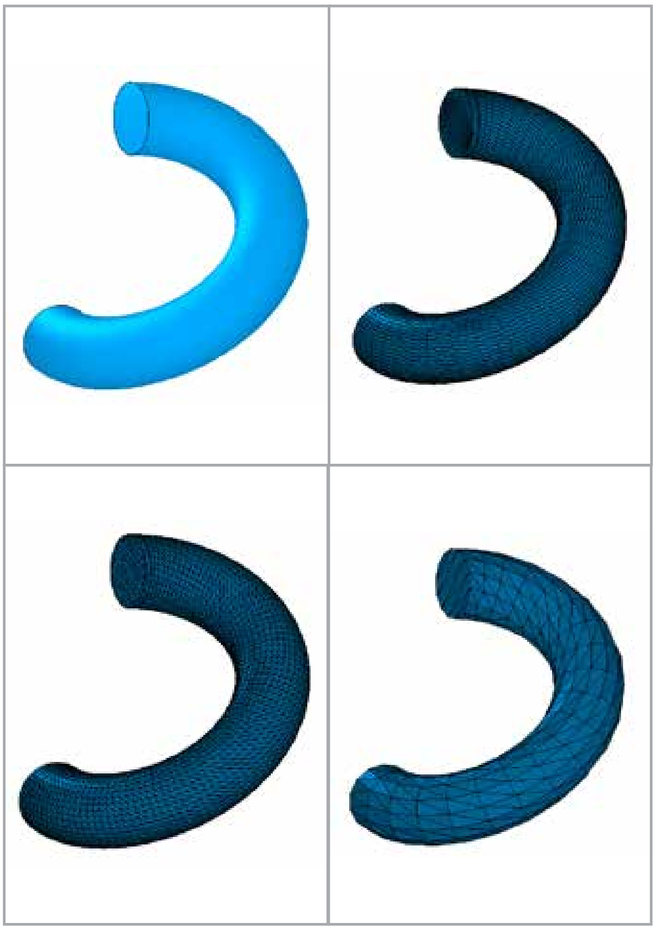


Fig.3 – Control of triangulation parameter input

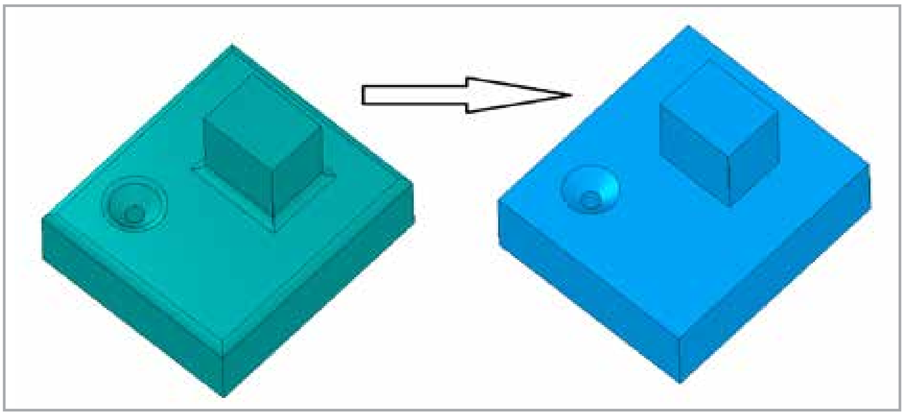


Fig.4 – Fillet removal in the C3D Modeler

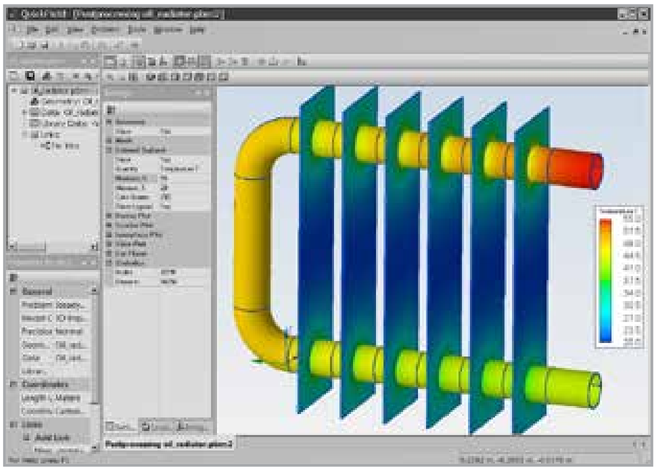


Fig. 5 - QuickField 3D from Tera Analysis based on C3D Toolkit

- *Identifying the contact faces of bodies with oppositely directed normals that have finite overlap areas.*
- *Constructing surface copies of NURBS with two-dimensional NURBS boundary curves based on the face of a shell."*

He continues, *"Today, several CAE developers use tools from our C3D Toolkit in their preprocessors. APM Research and Software Development Centre develops software for engineering analysis including strength, dynamics, continuum mechanics, and thermophysics.*

The scientific institute of Rosatom Corporation (RFNC-VNIIEF) uses C3D for complex Multiphysics modeling. Tera Analysis solves electromagnetic, thermal and mechanical problems with numerical simulation. PSRE Co. performs piping and equipment designs, as described in their case study."

Case study: preparing piping and equipment designs

PSRE Co. develops smart simulation and sizing software for piping and equipment design.

In 2013, the company licensed C3D Toolkit to enhance its PASS/EQUIP application, which comprehensively analyzes structural pressure vessels. The C3D kernel optimizes the geometric modeling functions and the computation of the properties of complex geometric shapes, such as their volumes, surface areas, centers of gravity, and inertia moments.

The powerful modern CAD kernel also expanded PASS/EQUIP's ability to export the geometric properties of vessel computing models to standard geometric formats. This allows PASS/EQUIP to provide tighter integration with popular CAD software.

Alexey Timoshkin, CIO of PSRE Co, comments, *"PASS/EQUIP Vessel is an application for calculating the strength of vessels and apparatus. It uses sets of equipment elements that are mainly calculated using analytical methods. These elements are included in the compound models of vessels.*

The calculation of the overall product consists of calculations of the individual elements according to the international codes, as well as calculations of interactions between structural elements. For these, numerical methods are used, including the finite element method (FEM). We chose C3D Toolkit because it is designed to create arbitrary models. This allows us to model equipment more accurately."

He continues, *"Some parts of the model require complex Boolean operations, such as modeling the tube arrays of heat exchangers. It is also necessary to calculate the mass-inertia properties of the*

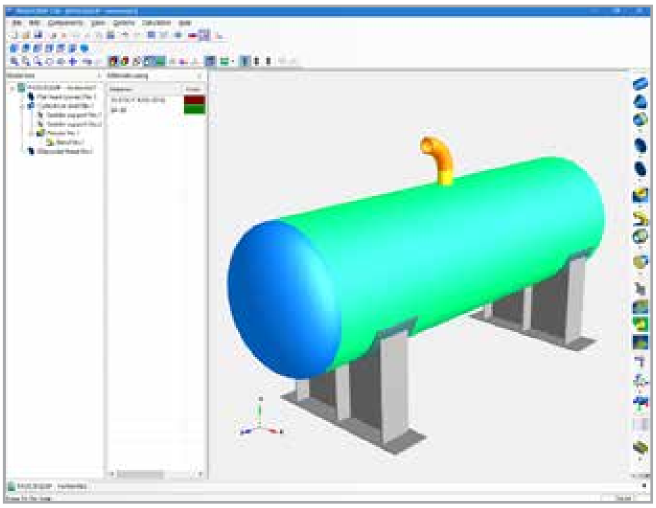


Fig 6. - PASS/EQUIP Vessel based on C3D Toolkit

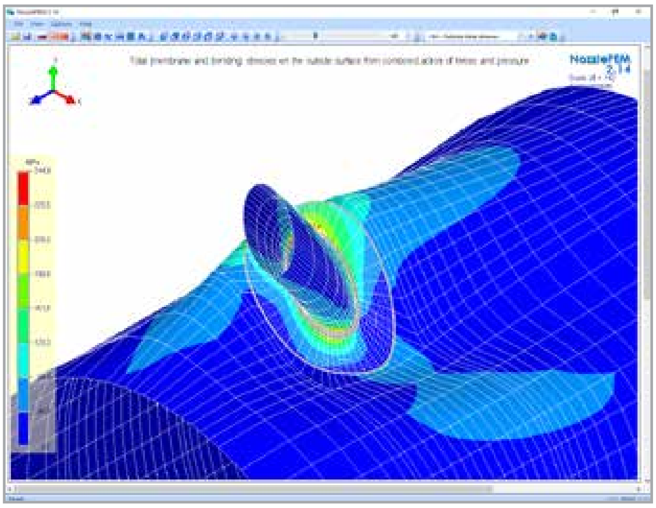


Fig. 7 - PASS/EQUIP Nozzle-FEM based on C3D Toolkit

model, including filling arbitrarily shaped cavities. The preprocessor has to solve geometric problems involving the surfaces of the shells, including determining the distances between fittings, finding the weakest sections of model elements, and so on".

"As a rule, these models are quite complex, especially the models of column-type apparatuses and heat-exchanger equipment, and so the speed of the geometric operations is very important to us. Thanks to C3D, we were able to eliminate a significant part of the geometry processing code and focus on high-level tasks instead," he states.

"In 2018, PASS/EQUIP Nozzle-FEM became the second software product from our company to use components from C3D Labs. PASS/EQUIP Nozzle-FEM uses the finite element method to calculate nozzle-to-shell junctions for equipment, as well as at tees and in equipment with complex geometry".

"These models quite often have complex shapes, such as junctions with toroidal-shaped inserts. These geometric elements are typically described using simple planes and quadric surfaces, but when they mating, complex fourth-order surfaces appear. The precision of the 3D model is key, as it directly affects the accuracy of the calculations. The geometric kernel is responsible for constructing the exact geometry in the model," he explains.

"We have been using the C3D kernel in PASS/EQUIP Nozzle-FEM since version 2.15, and are now using the current version, 3.1," he continues. *"The kernel stores curves and constructs intersections and fillets. The resulting shells are the basis for calculating the polygonal meshes required for calculations using FEM."*

"Initially, PASS/EQUIP Nozzle-FEM had predefined junction types that were previously modeled manually. Thanks to the C3D kernel, we were able to replace the predefined models with arbitrary ones. The current version of our software already contains twice as many different calculation models than before we deployed C3D. In future versions, we plan to provide specifications that are even more accurate and closer to real products. This will enable us to more accurately define equipment cases, calculate parameters for closely spaced nozzles, and combine arbitrary model elements" Timoshkin says.

"We use C3D not only for modeling, but also C3D Converter for exporting models to various CAD formats. C3D Converter has enabled us to increase the number of formats we support without needing to maintain our own data conversion code," he specifies.

"I would also like to mention C3D Labs' Help Desk. Their staff always help us quickly, are patient with our questions, open to discussion, and listen to our requests. This means we always receive the right solution and that we are able to influence improvements to the C3D kernel," Timoshkin concludes.

About C3D Labs

C3D Labs specializes in the technology-intensive market of software components. The company was founded in 2012 by the ASCON Group, a well-known CAD and PLM developer in Russia. Development of the C3D kernel began in 1995.

C3D Labs today provides its C3D Toolkit to engineering software developers for constructing and editing precise 3D models. As a tool for software developers, C3D Toolkit is the only package to incorporate the five modules critical to CAD: 2D/3D modeling, 2D/3D constraint solving, polygonal mesh to B-rep conversion, 3D visualization, and data exchange.

For more information
Oleg Zykov - C3D Labs
oleg.zykov@c3dlabs.com

By Eduardo Gª Márquez
Ingeciber

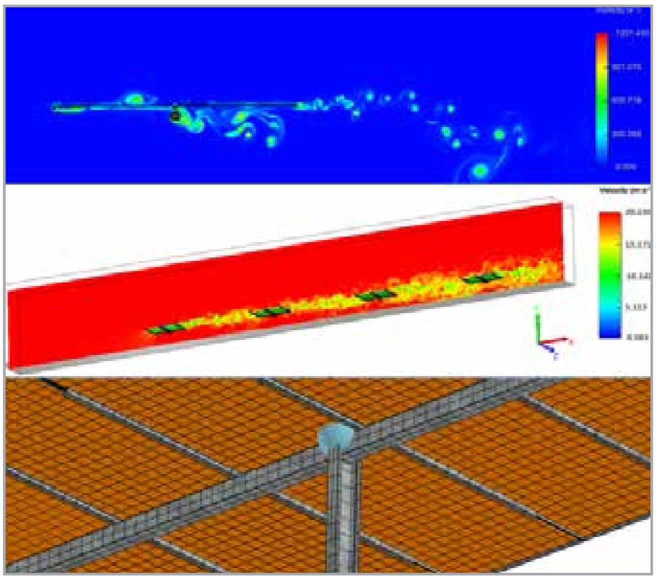
CFD and FEM enable analysis of wind effect on photovoltaic plants

Current optimized state of solar trackers requires the most advanced engineering design

Renewable energies are becoming increasingly efficient and competitive. High capacity for modernization, the increasing know-how among manufacturers and high competition in the sector are leading to a new generation of more efficient and affordable solar trackers.

As a result, more competent trackers are being manufactured at a much lower price than a decade ago, making photovoltaic electricity production costs so competitive that they are raising solar energy to a preferential position in global production.

Underlying all this increased efficiency, meanwhile, there is a high degree of specialization and very advanced engineering development which, by means of new computer tools and specialization of the engineers, succeeds in increasing the competitiveness of photovoltaic energy year after year.



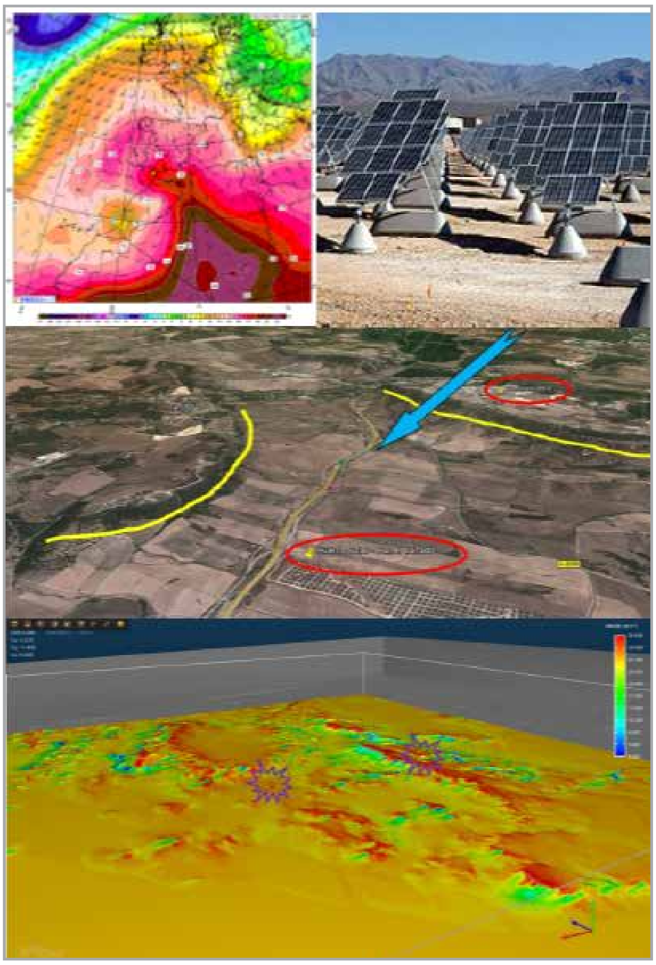
Wind interaction (CFD) with structural behavior (FEM)

These are the scenarios in which engineering and advanced consulting companies, such as EnginSoft in Italy or Ingeciber in Spain, offer complete solutions through advanced simulation, based on their extensive experience. Their engineers combine a high degree of specialization in different fields with the mastery of numerous fixed element method (FEM) and computational fluid dynamics (CFD) software.

Wind has been shown to be the biggest problem for photovoltaic electricity production today, since it has eliminated many solar trackers; wind CFD analysis is essential, therefore, not only to comply with standards but also to simulate the actual environment that these solar panels will experience.

Most CFD analyses are limited to studying the dynamic effects of wind (vortex shedding, galloping, battering). But CFD models can also be used to analyze how wind-breaking obstacles, such as fences or panels, help to reduce the impact of wind, and how wind actually behaves in the topography where the solar plant is located. This is done via the integration of data displayed by anemometers, with three-dimensional geometric models of the terrain, and CFD analysis, to provide engineers with the exact wind behavior over and above the standards and regulations of each region.

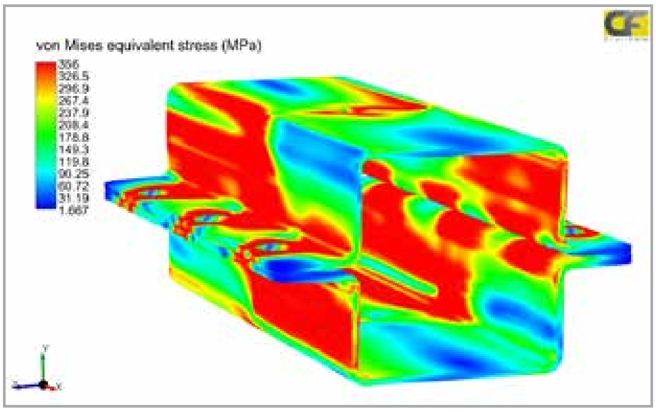
Precision and efficiency in this consulting is possible through the use of a variety of software that is suitable for combining dynamic fluid analysis and structural analysis using finite element calculations.



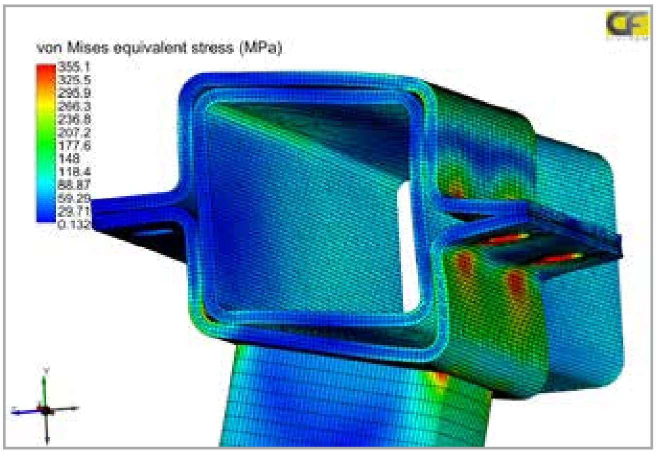
Wind action on the actual topography of the terrain

About Ingeciber

Ingeciber conducts simulations in many sectors mainly for the nuclear sector, wind energy, oil&gas, industrial components (ASME, pressure vessels, valves...) and all kinds of civil engineering.



CivilFEM model powered by Marc. Von Mises stress in a clamp. Customer STI Norland.



CivilFEM model powered by Marc. Von Mises stress in clamp. Customer STI Norland.

As a result, CivilFEM powered by Marc has become one of the most suitable FEM software applications for the structural analysis of solar trackers. It offers a solution to all kinds of necessary advanced analysis, from 3D modelling of all the industrial components of solar panels including those elements of special interest within the structure, the shaft connection elements or brackets (due to its different advanced contact types), to analyzing the interaction of wind (CFD) with the structural behavior (FEM). It can perform modal, static and dynamic analyses taking into account the inertial mass of panels and dampers, etc., and obtain the values of the natural frequencies of the structure, as well as the values of the deformations, rotations, efforts and stress of the structural elements such as the central axis.

For more information
Eduardo Gª Márquez - Ingeciber
e.garcia@ingeciber.com

Perfecting the manufacture of pressurized gas vessels using ESTECO optimization-driven design technology

The GASVESSEL project: lightweight optimization of a transportation system for compressed natural gas

By ESTECO

This article for non-technical readers discusses the optimization approach being used within the EU's GASVESSEL project to design super-light pressure vessels to transport compressed natural gas that have much higher payloads and dramatically lower transportation costs per volume of gas. The project aims to produce lightweight pressure vessels using filament winding, a popular method for manufacturing axisymmetric structures that are light and stiff. Using ESTECO's modeFRONTIER, the engineers were able to automatically evaluate, in few days, thousands of different designs that maximized the winding layer distribution and minimized their number while respecting the vessel's structural constraints, with the result that the first vessel prototypes, weighing up to 70% less, have already been manufactured and successfully tested.

Funded by the European Union, the GASVESSEL project aims to prove the techno-economic feasibility of a new transport concept for compressed natural gas (CNG). ESTECO, in partnership with other industrial organizations from the energy, Oil&Gas and naval engineering fields, has developed an innovative solution to manufacture pressure vessels that are considerably lighter than the current state-of-the-art alternatives. These super-light pressure vessels enable new ship designs that have much higher payloads and dramatically lower transportation costs per volume of gas.

CHALLENGE

Traditional pressure vessels normally used to transport liquified gas by ship cannot be used to transport CNG. This is because the relevant thickness of the ship walls required to maintain the operating pressure of 300 bar would add significant weight to the vessels, consequently reducing their loading capacity. In fact, one of the main challenges being addressed by the project is to produce lightweight pressure vessels for the transport of CNG using filament winding, which is a popular method suitable for manufacturing axisymmetric structures that are light and stiff. It involves the use of several layers of fiber-reinforced composite materials wrapped around a thin internal metal liner.

> *"The optimization process makes gas vessels up to 70% lighter."*

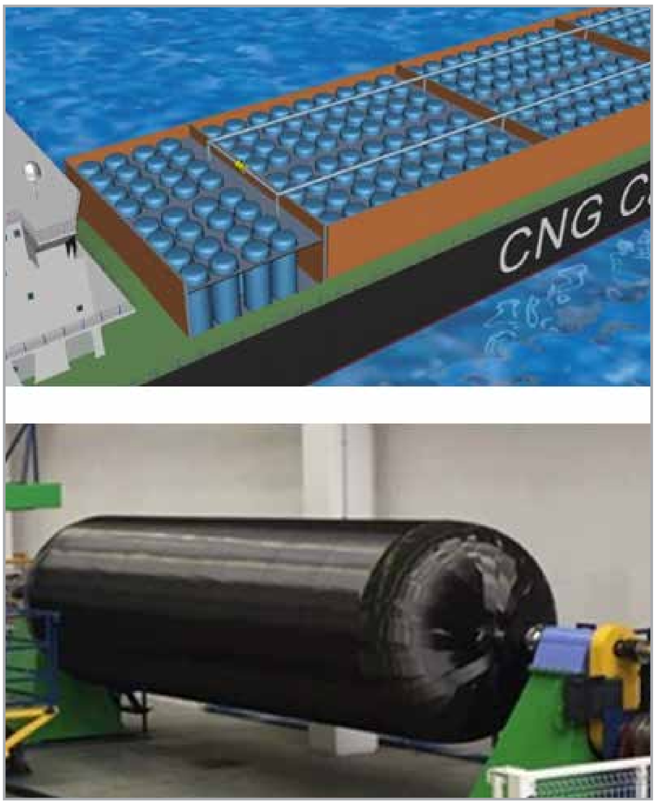


Fig. 1 - Gasvessel CNG carrier - Pressure vessels manufactured using filament winding

SOLUTION

During the design phase, the material and geometrical parameters of the vessel (mainly related to the number and winding angle of the layers, the percentage of composite fibers and the liner's mechanical properties) were considered for optimization to reduce the weight and costs while honoring the structural constraints. The winding process was physically modelled with CADWIND software to evaluate the distribution of composite layer thickness at each point of the vessel.

Fig. 2. The modeFRONTIER workflow for the winding optimization

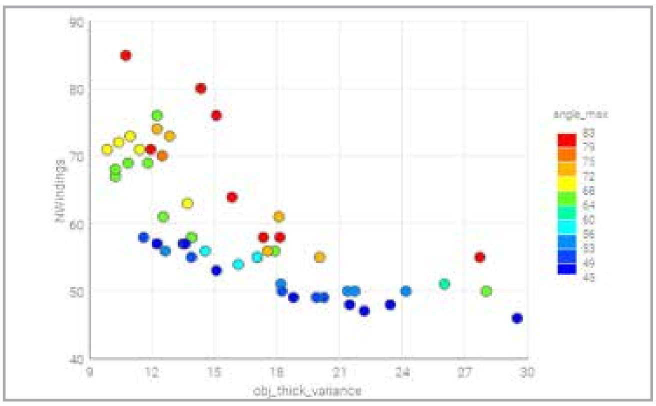


Fig.3. The optimization results: number of windings vs thickness – standard deviation [mm] and maximum angle [°] of winding

About ESTECO

ESTECO is an independent software provider, highly specialized in numerical optimization and simulation data management with a sound scientific foundation and a flexible approach to customer needs. With 20 years experience, the company supports leading organizations in designing the products of the future, today. ESTECO is the owner of modeFRONTIER, a comprehensive solution for process automation and optimization in the engineering design process.
For more information, visit: esteco.com

About the GASVESSEL Project

The key to securing Europe's energy supply is diversifying supply routes. This includes identifying and building new routes that unlock resources and decrease Europe's dependence on a single supplier of natural gas and other energy resources. The GASVESSEL project opens up new possibilities to exploit stranded, associated and flared gas where this is currently economically not viable and creates new cost-efficient gas transport solutions. This will be achieved with a novel offshore and onshore compressed natural gas (CNG) transportation system. For more information, visit www.gasvessel.eu

The project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 723030.

The filament winding simulation model and the stress analysis of the vessel were then integrated in a modeFRONTIER workflow to evaluate the different solutions and choose the best designs. The optimization task, which aimed to maximize the uniformity of distribution of the winding layers and minimize their number while respecting the structural constraints of the vessel, was conducted using pilOPT, ESTECO proprietary autonomous algorithm.

BENEFITS

modeFRONTIER process automation and optimization capabilities enabled the engineers involved in the project to automatically evaluate thousands of gas vessel designs in just a few days, as opposed to losing weeks by doing it manually. The Bubble Chart allowed them to visualize and identify the best candidate designs among those with the lower weight and manufacturing costs. As a result, the first gas vessel prototypes, which weighed up to 70% less than the vessels not reinforced with filament winding, could be manufactured and have already been successfully tested.

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

The scope of and potential for Italy's role in European research

An interview with the Agency for the Promotion of European Research (APRE)

By Kathleen Grant
EnginSoft

The newsletter met with Marco Falzetti, director of APRE, for some insight and perspective on the new Horizon Europe Framework Programme and the role that Italian research and business has played so far, as well as the potential for its participation in the future and how this can be expanded and increased.

Q. Can you start by giving us an overview of the research program in Europe and Italy's role in it?
MF. Let me start by saying that Horizon 2020 was, and Horizon Europe will be, the largest research program in the world. It characterizes Europe as an aggregate system more than any other great continental system. However, Europe's undertakings through the Framework Programme cannot substitute the activities that must be undertaken by the individual member states. In other words, although the Framework Programme is the largest systemic research program in the world, it cannot be expected to replace all of the individual European countries' research efforts.

It therefore follows that it cannot be the only solution to R&D needs in Europe, but it does have two considerable merits. The first is the economic aspect, which is, however, marginal in terms of the overall individual European research effort.

The second, and much more important, merit is that for twenty years now it has been bringing together national groups and accustoming them to working together in cooperative research efforts and activities. This signifies dealing with different cultures, learning to work with people who use different languages, but above all, being able to engage with the best minds. Undoubtedly the Framework Programme is extremely competitive, selective and high quality.

Coming to Italy, it is necessary to understand the national research situation. One thing that must be understood is that successful participation in this competitive European research effort requires and arises from the national research efforts, since these form a type of preparatory and complementary role that cannot be ignored. This national research component in Italy is much smaller than among other European countries of similar size. However, Italy does make a great effort to participate and also manages to achieve a relatively good economic return on this participation if measured in terms of the impact of its pool of researchers. Of course, in absolute terms, if we compare Italy to other, roughly similar-sized European countries, Italy could do even better.

Q. Are there any other factors that affect this?
MF. Another element that plays a role, I would say, are the administrative and bureaucratic structures underlying the work of researchers. We clearly know that an Italian researcher who may contemplate a role as the coordinator of a European project will be forced to assume a whole series of activities, responsibilities and commitments that are way beyond their duties as a scientific researcher.

Because they feel poorly supported by the administrative and organizational system that should be in place, they are discouraged from taking on certain more active and significant roles in many projects. This, in turn, means that the number of projects coordinated by Italy is, on average, lower than other countries, which essentially means that Italy participates, but in marginal roles.

From an Italian perspective, this choice of participating only in marginal roles often limits the scope of the project, as well as the ability to design the projects according to one's specific vision and profile, and also increases the necessity for compromise from the outset.

Q. Participation in European research is intended to and seems to improve innovation in Europe, but this does not seem to be the case in Italy. Why is that?

MF. Here you are referring to Italy's position in the European Commission's Innovation Scoreboard. Just to be clear, Italy has always been more or less in the middle of that chart, i.e. in a position that is neither too dramatically poor, nor one of the top five. The reason for this is a bit more complex and is not only related to research, but also to Italy's industrial environment. While there are certainly many examples of great innovation in Italy, one must also remember that a substantial part of the Italian entrepreneurial sector requires research for incremental innovation – innovation in existing processes to make them more competitive, efficient and environmentally friendly, to improve product quality and performance, and so on.

So, you could say that we were forced to fight a defensive battle in the European Innovation Council when the Commission was pushing very hard for so-called disruptive innovation, as opposed to other forms of somewhat more traditional innovation, which one could also call incremental innovation. We knew Italy had to fight this battle because our industrial system requires this type of important and crucial innovation support. Another aspect is that a large part of Italy's GDP today comes from production activities generated by SMEs, and we have to increase the innovative capacity of these existing businesses and industries, and not only focus on innovative start-ups.

When we talk about innovation, what matters is dialogue and collaboration between those who generate and archive knowledge, so research and academia, and those who are able to transform that new knowledge into economic value, which tends to be the business or entrepreneurial world. Despite the many examples of robust and stable collaboration that exist, in particular in some areas of the country, we can still improve this at a systemic level while also focusing on the excellence that does exist among SMEs, which are already very future-orientated and already operate in tune with the times.

Q. Are there any other factors or challenges in Italy in this regard?

MF. Perhaps you could say that in other countries the famous link that they call the transition from the Valley of Death works more and better than it does in Italy. This would be the link between the generation of the innovative knowledge and technology and its exploitation in the market. It is not that this capability does not exist in Italy; it just seems to work better elsewhere, possibly because creating a new business here is more complicated than in other European countries, for example.

This goes back to the subject of the bureaucratic and administrative system in Italy. Even though much has been done to simplify it, it is still much simpler to open a start-up in the UK near Oxford, for example, than to open a start-up in Italy in all things innovative. If you compare two systems, one in which entrepreneurial risk-taking has been facilitated and simplified for many years, and ours, it is clear that it is still more complex to do in Italy and so there is less willingness to do so and more resistance to risk-taking than there is in other countries where they have already been doing it in a simpler, more aggressive and more dynamic way for many years. So, this has to do with a bias that has very little to do with the culture or ability of Italian R&D, and much more to do with the administrative machine around it.

Q. Some researchers comment that while participating scientists are strongly encouraged to patent their ideas, the process is complex and difficult, and help to do so is lacking. What is being done in this regard in Europe?

MF. The answer here is a bit more complex. If we stay within the Framework Programmes, the European Commission has always been attentive to the intellectual property rights (IPR) issue – from as early as the fifth programme – and has consistently tried to raise awareness of the importance of scientists defending their IPR. There is, however, a cultural dimension to this issue among researchers and scientists. Let me explain. In the mind of many researchers and scientists, there is a tendency to consider the highest valorization of their work to be the act of scientific publishing.

Clearly, this is not the best mind-set if you are wanting to secure and protect your work. It is not enough to tell the researcher not to publish because publishing naturally precludes the possibility of applying for a patent or doing other things to defend yourself in some way.

I can say that in the research activities funded under the Framework Programmes, the Commission has always paid appropriate attention to the IPR component. It is very different when we discuss those researchers or project coordinators that already have a careful eye tuned to the entrepreneurial dimension, because they undertake the activities with a clear understanding of what they have to worry about in terms of IPR, too. So while it is probably true that a researcher may find it difficult to deal properly with the IPR dimension, it is also true that there are many systems at both national and European level that are there to give them a hand in understanding what it means to defend the IPR of their work.

Q. This fear is becoming greater now with the current proposals to expand participation to other countries, some of whom are obviously commercially shrewder. Are there any initiatives underway to protect it more?

There are some countries that are quite aggressive in the way they approach the issue of competition and this is definitely a matter for concern. I have to say this: Europe has been very open to the outside world, perhaps not caring enough about protecting itself in this extreme openness. So, I would say that European ideas have certainly been a little more exposed to cannibalism than in other systems. Europe apart, even in my experience, it has often happened that in a meeting the Europeans talk and share a lot and there are many others who listen, but these other countries in fact say almost nothing. So, this picture of a Europe that has nurtured openness and has received very little in return in spite of giving so much, is certainly not wrong. Why this has happened, whether because we were naïve, or because we are more open-minded and collaborative, I leave to you to judge.

What is happening now is that this is finally beginning to be assessed more carefully. Horizon Europe will certainly continue to be a widely inclusive program, but much of the discussion that is still open on the new program specifically concerns the mechanisms and rules of access for the famous "associated states". These rules for association are currently being defined, but are likely to allow anyone to participate as long as they respect the European rules of sharing, participation, inclusivity and transparency in the work, while there may be a need to block or stop participation by systems that potentially do not share the same fair and open-minded approach.

When we look at the third pillar of Horizon Europe and the so-called European Innovation Council, whose activities are aimed not so much at funding collaborative research as on developing activities to support new entrepreneurs or on advanced entrepreneurship – activities that are tendentially mono-beneficiary to support the developing and bringing to market of certain knowledge – there are discussions underway about whether it would be appropriate to open this part of the program to any state. It is important to remain open in all the collaborative research activities, while securing European scientific and industrial interests.

Q. How can companies and researchers be more successful with their proposals?

MF. From an Italian perspective, there is definitely a quality issue. We need to improve our ability to produce proposals that are well written, where the ideas are clear, the partner component is correctly constructed, and so on. This quality can be improved by improving the ability to write, to linearize a logical process, to describe it, and to quantify its financial aspects correctly. We know what has to be done. The other element, however, is understanding how best to assist with this improvement process. In a program that has become increasingly selective, challenging and high quality, it is unreasonable to think that everyone's performance can be improved in the same way. This is unrealistic and is a bit like saying that we have to take everyone to PhD level in one shot – irrespective of whether they are primary school students, middle or high schoolers, or university graduates. It is just not possible.

So, we have to have at least a two-tiered approach. First, we have to work on the excellence that already exists at a national level. There are entities that currently are well prepared, who already have good or excellent knowledge of the Framework Programme, are able to write well in a second language, know how to express complex logical processes very well because they are used to reasoning in a certain way, and know the rules of engagement. These are the candidates that have the greatest possibility of achieving good results in the shortest time. This is very high-profile training because they are in the last part of the curve and so achieving an improvement requires a lot of energy because of the complexity and refinement entailed.

Secondly and in the meantime, we also need to continue working on a larger spectrum of national applicants with a different campaign that takes a longer time. This is more of a mass approach to improving their Framework Programme literacy, helping them to understand the possibilities of the system, and helping them along a path that does not last one or six months or a single call, but must last a few years to bring them to a competitive level.

Q. Which are Italy's areas of excellence in the Framework Programmes?

Italy already excels and has a huge capacity to excel scientifically, technologically and industrially in many areas of the program e.g. transport, industrial technologies, energy, SME. Paradoxically, there is no country whose industrial sector wins, in relative terms, as much as Italy does. Percentually, our industrial sector nationally wins a larger sum of money in the Framework Programmes than any other European country's industrial sector – including Germany.

This means that Italian industrial participation in the Framework Programme is outstanding compared to other competitors. However, this also needs to be interpreted segment by segment

About APRE

APRE, the Agency for the Promotion of European Research, is a non-profit research organization.
For over twenty-five years, APRE, in close collaboration with the Ministry of Education, University and Research (MIUR), has provided its members as well as businesses, government agencies, and private individuals, information, support and assistance for participation in national and European programmes and collaborative initiatives (today, with particular reference to Horizon 2020) in the field of Research, Technological Development and Innovation (RTDI) and in the transfer of research results.

APRE was created in 1989 as a joint initiative of the Italian Ministry of Education, University and Research (MIUR) and some public and private bodies in order to meet the growing demand for information on European research programmes. First reality of its kind, APRE has been supporting the scientific and the industrial community for over 20 years in the path to Europe, and today, to the World, through offering information, training and assistance activities on the participation rules of the Framework Programme of the European Commission.

APRE is supported by more than 100 supporting and ordinary members
APRE Members are public or private bodies that share the institutional mission of the Agency and at the same time are an active and beneficiary part of the activities carried out by APRE.

Public and Private Research centres, Universities, Scientific Parks, Public Administrations, Trade associations, Finance, Chambers of Commerce, Technological Districts, enterprise, other.

EnginSoft has been an ordinary member of APRE since 2019.

because the situation is not the same everywhere. This can lead us to two strategic choices: one is to try to improve everything incrementally across the board and aim for maximum performance, the other is to concentrate our efforts on those traditional areas where Italian research and industry evidently excels in the expression of its know-how.

Know-how is not just a matter of industry. Obviously, it includes scientific knowledge, too, because it goes without saying that excellent industrial performance in a specific sector must be informed, sustained and supported by great scientific knowledge – whether that be in informatics, micro-electronics, advanced materials, advanced mechanics or robotics is immaterial at the moment. So, when I say that there are enormous industrial capabilities in Italy, it signifies that there is also significant academic excellence in those areas which advances and transfers the knowledge in that sector, as well as basic research that remains a nurturing element of the knowledge creation process. It is all inter-related and inextricably linked.

Q. Some argue that companies benefiting from this type of public investment do not survive long after the end of that investment. Can you provide some clarification on this, especially with regard to SMEs?
MF. Possibly for the very reasons I was outlining before, Italian SMEs have tended to use these European investments to innovate in areas that they may not have innovated in before, specifically because of a lack of resources. So, tendentially, Italian SMEs tend to have had an entrepreneurial existence prior to the research projects, and they tend to apply for funding for incremental innovation rather than for the innovation that is more typical among start-ups. You can say that the Italian companies pre-existed the financing and therefore continue to exist afterwards, too. The European financing for innovation research thus helps enormously in providing resources to assist their growth in innovation terms.

Your question is more linked to the new generation of entrepreneurs, born as a result of the European financing for start-ups, who mainly tend to struggle as soon as the financing ends. There are many examples of start-ups that began and developed well under the investment programs, but that then start to struggle or fail as soon as the funding ends. This means that it is important not only to invest in the launch of new entrepreneurial initiatives, but also in creating the conditions – the famous "ecosystems for innovation" – in which innovation comes to light and then manages to grow and succeed.

Just how much of this future growth and success should be the responsibility of the public system that helped to birth it, and how much should be the responsibility of these entrepreneurs and the system to assist them to then make it on their own, is open for discussion.

However, the Commission is considering the question. If you consider the third pillar of the future Horizon Europe program, you find the European Innovation Council (EIC) not only as a dedicated instrument for creating entrepreneurs, but also an entire structure called "ecosystems for innovation" that, while they still have to be given a detailed form within the program, are there to construct/create the enabling conditions to aid these seedlings that have germinated within the ambit of the activities of the EIC to find adequate support that enables them to grow autonomously in an environment that is highly competitive and open. So, while these seedlings need to be watered attentively and need a terrain that nurtures their growth, obviously their ultimate success depends on them.

This European innovation ecosystem is currently being designed and created to generate systemic capabilities within both the start-ups that have been incubated within the EIC, and the others that were established elsewhere.

EU research funds advances towards neuroprosthetics and builds multidisciplinary capacity

Greater protection of EU IP is necessary before increasing non-EU participation

By Kathleen Grant
EnginSoft

Prof. Stefano Vassanelli is the Associate Professor of Physiology and head of the NeuroChip Laboratory at the University of Padua's Department of Biomedical Sciences. He is also currently involved in two EU projects in the Horizon 2020 (H2020) Future and Emerging Technologies (FET) Open programme, namely SYNCH, which he coordinates, and NEUREKA which arose from a proposal he drew up. The H2020 FET Open programme supports the early stages of the science and technology research and innovation around new ideas towards radically new future technologies. In this issue, the Newsletter spoke to Prof. Vassanelli to understand a researcher's view on the value and contribution of the EU research undertakings.

Q: Stefano, please tell us a little more about the two projects you are involved in

SV: There is a common denominator in both projects, which is to make cerebral neurons communicate physically using a form of communication that emulates neurological communication. In other words, to create a hybrid system where synthetic and biological neurons communicate. It is, essentially, a new way to think about the human-machine interface: electronic neurons in a network of hardware and software read the brain's signals and respond using an artificial intelligence-based method of calculations.

Synch (SYnaptically connected brain-silicon Neural Closed-loop Hybrid system) is a hybrid approach using artificial neuronal networks emulating real brain networks to elaborate brain signals, and to simulate the brain adaptively. It aims to develop an advanced method of stimulating the brain, working as a form of "cerebral pacemaker", in particular to stimulate certain deep structures of the brain. At the moment, deep brain stimulation consists of electrodes connected to an external stimulation box and a neurologist has to adjust the signals, so the system is still very primitive, if you will. This project aims to read the signals coming from a brain with a neuropathology (e.g., Parkinson's) and then, using artificial intelligence (AI), to reset a normal neurological activity. There are obviously applications for Parkinson's disease, but also for the treatment of epilepsy and for stroke rehabilitation. The aging population and improved diagnostic methods have led to an increase in the prevalence of these pathologies and this project uses electronic stimulation targeted at providing a supportive therapy to patients in addition to their pharmacological therapy, particularly since pharmaceutical companies are increasingly switching research investments away from these drugs and to other areas of research that are less complicated.

The NEUREKA project instead connects at a nanoscale. There are nanoelectrodes in a chip that try to attach to neurons to create a high-resolution interface. Today it is possible to measure neuronal activity, such as a neuronal spike, or other neuronal outputs, but we do not know how to stimulate neurons at the dendrite level, where the synapses act. When the synapses malfunction, like in Alzheimer's disease, we don't have way of directly modulating the synaptic responses in the dendrites in many neurons simultaneously. In this project, we want to simulate the synaptic inputs at dendritic level through nanoelectrodes that are intimately integrated in vitro with neurons growing in culture.

In brief, this project is aiming at creating a hybrid electronic-human "lab on a chip" to help us to better understand what happens in pathologies like Alzheimer's and to then use this hybrid for drug development and discovery. This model will help us to further improve our ability to understand and test our predictive models for Alzheimer's.

Q: What advice do you have for others about creating and coordinating EU projects successfully?

A: It is important to develop experience by becoming a reviewer for other EU projects. It also requires crafting proposals, seeing

them rejected and learning from your mistakes. Participation is key.
Technically, as the coordinator, you need to be convinced about the project, or it won't go well. The work is important. With regard to the creation of consortia, you need to be in a position to choose project partners that have one common goal. These partners must be chosen carefully and well based on their being the best fit for the project objectives. Then, as coordinator, you need to strive for excellent collaboration with a view to building a network that becomes an ecosystem to feed and support the project outcomes by combining the best endeavors of all the partners.

Q: What are some of the current issues in the European research environment?

SV: The suggestion of increasing the aperture of and participation in EU research projects to other non-EU nations is a double-edged sword, in my opinion. Obviously, on the one hand, these other countries add resources and competence, but unfortunately, these external non-EU partners also represent the significant risk to the European participants that they will exploit the know-how developed by years of European investment. One important issue is that many EU systems and institutions do not have a commercial culture nor the infrastructure or the capacity to effectively and widely communicate the results of their projects.

In many cases, these external partners are expert at this and so they are able to structure the communications in such a way as to exalt their participation in the projects and to spread the news of the discoveries as though these were theirs.

While there is encouragement at the EU level to protect the intellectual property (IP) of EU researchers, I believe there is a need to do more at a political level to protect this IP. At the moment, researchers are told to patent their ideas, but this process is difficult, time consuming and expensive. There should be a monitoring organism that tries to select the best opportunities and to protect those ideas. We need a greater culture of protection and guardianship and we need to be very careful to safeguard the EU know-how that we've been investing in and developing over the past 30 years.

The new vehicles of financing via new investments, angel financing and venture capital are a positive thing and should be boosted. But it is vital that this form of investment impetus does move in a direction that is to the detriment of fundamental research. The exploration into visionary scenarios is vital and very European and it has created a very innovative ecosystem, a domain of human resources, skills and capital.

Q: What have EU projects accomplished from a research perspective?

SV: These two projects, like almost all the other EU projects, are multi-disciplinary and multinational. The principle behind the idea is to create a single unified team of researchers distributed across Europe in order to access resources and competencies that wouldn't otherwise be available. From that perspective, these EU FET projects represent an exceptional programme. This type of exploration would be very difficult to conduct in other projects. There are some specific challenges at the moment, for instance Brexit is posing some important problems for both existing and future projects because it is unclear how the involvement of British scientists and researchers will be resolved.

The collaboration between the business and academic worlds is very important across the EU, too. The participating companies have a lot of specific competencies and knowledge that they bring to the table, as well as knowledge and experience of market demand, so one could say that they bring the academic world towards reality. But, there are also negative aspects to this because, often, the academic partners lose out on the commercial benefits of the projects. I believe that this model needs to be readdressed slightly so that the academic world is not defrauded of its ideas by not realizing any profitable benefits. It is essential to create more of a virtuous circle for the universities.

In the academic sector, there are national realities like the Italian one, but it is not the only one, where it would be impossible to do some of these types of research because the different disciplines are kept very isolated, avoiding multi-disciplinarity.
This tends to funnel researchers into very specific channels, creating super specialists.

While super specialists are indispensable, so are people who see the big picture; people who have the ability and transversal knowledge to understand and translate what specialists do. Addressing the problems and challenges of the future requires us to build people who have multidisciplinary expertise. In Italy, the creation of multidisciplinary profiles doesn't exist. This is largely a result of cultural fact, because the academic approach is still a bit old-fashioned: each person moves in the narrow world of their skills and knowledge. This is aggravated by the fact that there is the political question in academic circles of "controlling" a discipline, so agreeing on sharing resources is almost impossible.

While it seems that there are similar problems in companies, they have the advantage of greater flexibility and the habit of responding to market pressures. The academic world is different. So, these EU projects have helped to break down these barriers between departments and disciplines, because the projects involve many interdisciplinary fields. This is a strength of the EU research investments that has created, and which continues to nourishes a new dimension in both business and academia. From that perspective, these investments are not wasted money. I am convinced that they have helped to create some unique skills in Europe.

Finally, I have to emphasize that my career would have been impossible without these European projects.

SYNCH project: improving neurostimulation implants

EnginSoft uses FEM techniques to model micro capacitive needles to predict electrical behavior after implantation

By Giovanni Falcitelli
EnginSoft

SYNCH is a European research project within the framework of the Horizon 2020 – Future and Emerging Technologies programme (Grant agreement ID: 824162). It is the acronym for SYnaptically connected brain-silicon Neural Closed-loop Hybrid system.

The scientific and technological objective of the project is to create a hybrid system where a neural network in the brain (BNN) of a living animal and a silicon neural network of spiking neurons on a chip (SNN) are interconnected by neuromorphic synapses, thus enabling co-evolution of connectivity and co-processing of information of the two networks.

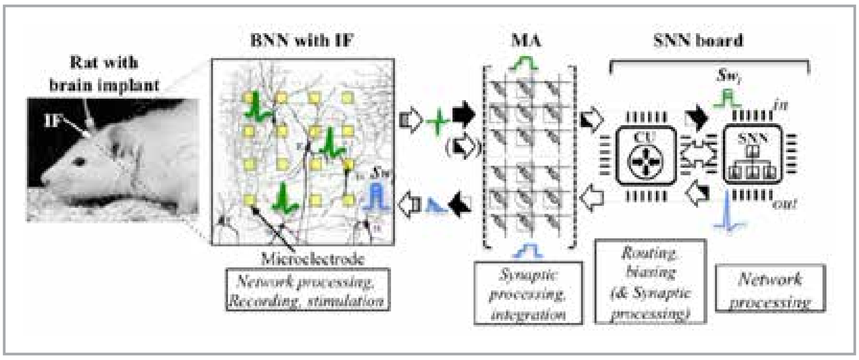


Fig. 2 - Overall methodology for concept implementation

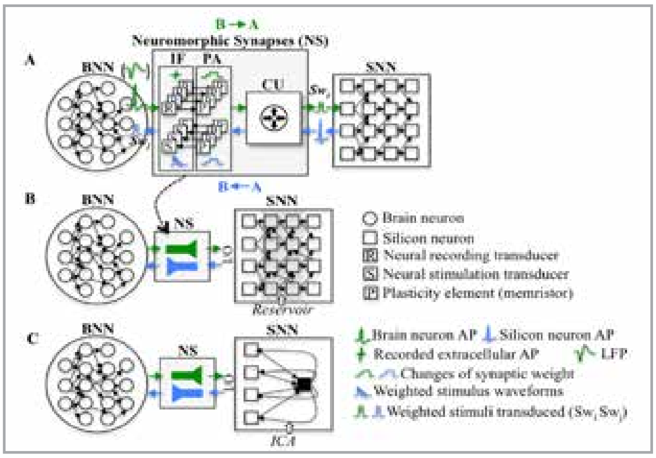


Fig. 1 - A The technology concept; B. The SNN implements reward-based learning by reservoir computing; C. The SNN functions as a non-linear Independent Component Analysis (ICA) processor

The aim is to establish a reciprocal link, inspired by synapses, between a neuromorphic chip of silicon-based spiking neurons and an in vivo brain network. Thus, we want to use the SNN as a processing architecture to adaptively stimulate and rescue functionality in an animal model of disease. In essence, this concept envisages an invasive brain-computer interface in which brain-inspired processing replaces standard PC-based computation. In this hybrid system, brain-inspired processing occurs at two different levels of complexity, in two separate neuromorphic devices.

Memristors (memory resistors) operate synaptic-like processing, linking artificial and biological neurons by emulating synaptic integration and plasticity. Worth noting is that, in future, these single devices may form part of implants designed to rescue simple reflex-based circuits (e.g. localized in integration centers of the spinal cord or brainstem) for example to treat autonomic nervous system dysfunctions. A higher level of processing occurs in the SNN, where brain-inspired computation relies on neuronal network dynamics. We envisage that, either alone or in combination with memristor arrays, the SNN may become part of neuroprostheses to treat focal pathologies in higher brain structures such as the cortex (e.g. in epilepsy or stroke), or in the basal ganglia (e.g. in Parkinson's disease). It is worth noting that, despite the tremendous advances in brain-inspired computation with artificial neuronal networks, theoretical work is key to revealing the innovative potential of this approach.

EnginSoft's contribution

The NeuroChip laboratory of the Padua Neuroscience Center (PNC, http://pnc.unipd.it) is headed by Dr. Stefano Vassanelli (http://www.vassanellilab.eu). As part of the research to improve neurostimulation implants, the PNC entrusted EnginSoft with the development of a full parametric finite element (FE) model of a microelectrode array in order to predict their electrical behavior once implanted in brain tissue.

The microelectrode array represented is an assembly of ten units, each of which is equipped with one stimulation plate and several sensing microelectrodes.

This has three innovative elements compared to the current technology:

1. The stimulation plate and the electrodes are electrically insulated from one other in order to physically separate the channels for the signals used to stimulate the neural tissue from the signals coming from the neurons.
2. The transmission of electrical signals occurs by capacitance much more than by conductivity.
3. Iridium Oxide (IrO), which is the standard material used for microelectrodes, is replaced by Titanium Dioxide (TIO2).

EnginSoft used the Ansys Electronic Desktop to perform the challenging task of creating a parametric FE model of the capacitive needle containing all the microelectrodes submerged in a conductive dielectric with properties similar to neural tissue.

The main software used was Maxwell3D owing to the following useful features:

1. Ability to easily build robust parametric geometries from scratch within the ACIS modeler
2. Auto-adaptive meshing strategy useful for ensuring the desired accuracy of the results
3. High-performance computing solvers to solve million DOF (Degrees Of Freedom) FEM models
4. Powerful post-processing tools for plotting contour and vector representations of electric potential, electric field and current density
5. Automated routines to calculate the capacitance matrix and the conductance matrix

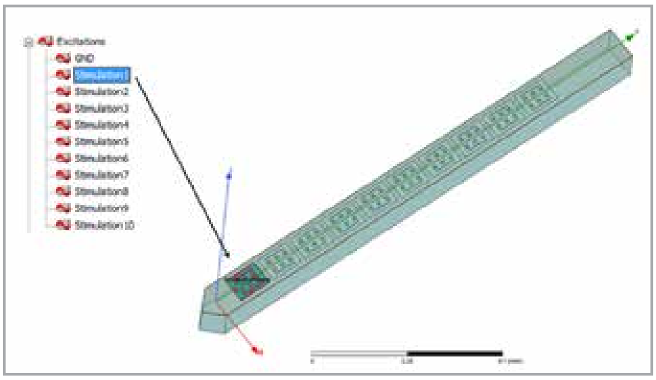


Fig. 4 - Excitation setup to obtain Capacitance and Conductance matrices for stimulation regions

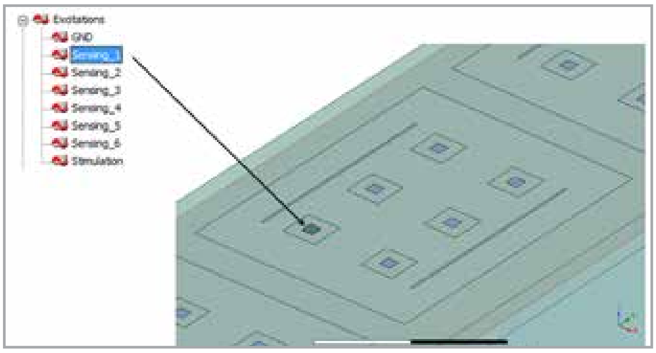


Fig. 5 - Excitation setup to obtain Capacitance and Conductance matrices for sensing regions

Name	Value	Unit
d1	18.5	um
d2	17.25	um
d3	37	um
d4	82.8	um
d5	150	um
h_Si	142.7	um
h_SiO2	7.3	um
h_Ti	50	nm
h_TiO2	30	nm
Lr_electrode	90	um
Ls_electrode	5	um
Lsensor	133	um
Rel	0.90	mm
Wr_electrode	1	um
Ws_electrode	5	um
Wsensor	100	um
Wshaft	138	um

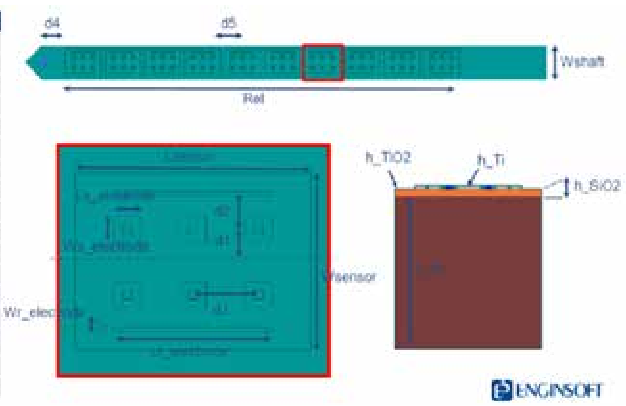


Fig. 3 - New design of capacitive microelectrode array for neurostimulation implants

To study any electrical coupling effect between the electrodes, two analysis scenarios were established:

- Setup 1: 10 independent potential sources were defined to evaluate the electrical current density, capacitance and coupling between the stimulating electrodes. (Fig. 4)
- Setup 2: 7 independent potential sources were defined to evaluate the electrical current density, capacitance and coupling between the stimulating electrode and the sensing plate. (Fig. 5)

Solving these two scenarios provides the capacitance matrix and the conductance matrix necessary to predict the electrical behavior of the microelectrode array once implanted in the brain tissue.

- Typical values obtained for scenario 1 are:

Analysis Setup 1

Capacitance Matrix [pF]

	Simulation1	Simulation2	Simulation3	Simulation4	Simulation5	Simulation6	Simulation7	Simulation8	Simulation9	Simulation10
Simulation1	0.25899	-0.088657	-0.024987	-0.019203	-0.016592	-0.015356	-0.014499	-0.01402	-0.013995	-0.017221
Simulation2	-0.088657	0.29306	-0.081661	-0.019964	-0.014875	-0.013063	-0.012045	-0.011509	-0.011415	-0.013993
Simulation3	-0.024987	-0.081661	0.29551	-0.080618	-0.018889	-0.014313	-0.012533	-0.011703	-0.011475	-0.013977
Simulation4	-0.019203	-0.019964	-0.080618	0.29431	-0.077958	-0.018721	-0.014121	-0.012518	-0.011995	-0.014437
Simulation5	-0.016592	-0.014875	-0.018889	-0.077958	0.29115	-0.077788	-0.018538	-0.014155	-0.012881	-0.015142
Simulation6	-0.015356	-0.013063	-0.014313	-0.018721	-0.077788	0.29427	-0.079806	-0.019025	-0.014936	-0.016659
Simulation7	-0.014499	-0.012045	-0.012533	-0.014121	-0.018538	-0.079806	0.2956	-0.080169	-0.018863	-0.019105
Simulation8	-0.01402	-0.011509	-0.011703	-0.012518	-0.014155	-0.019025	-0.080169	0.29484	-0.081263	-0.024882
Simulation9	-0.013995	-0.011415	-0.011475	-0.011995	-0.012881	-0.014936	-0.018863	-0.081263	0.29336	-0.088553
Simulation10	-0.017221	-0.013993	-0.013977	-0.014437	-0.015142	-0.016659	-0.019105	-0.024882	-0.088553	0.28801

Conductance Matrix [mSie]

	Simulation1	Simulation3	Simulation4	Simulation5	Simulation6	Simulation7	Simulation8	Simulation9	Simulation10	Simulation2
Simulation1	0.51274	-0.056074	-0.044818	-0.039971	-0.037443	-0.035843	-0.035065	-0.035367	-0.043801	-0.18436
Simulation3	-0.056074	0.50972	-0.16647	-0.042516	-0.033461	-0.030094	-0.028628	-0.028472	-0.034986	-0.16901
Simulation4	-0.044818	-0.16647	0.59101	-0.16493	-0.04192	-0.033044	-0.030096	-0.029382	-0.035767	-0.044577
Simulation5	-0.039971	-0.042516	-0.16493	0.59051	-0.16463	-0.041776	-0.033355	-0.031187	-0.037247	-0.034898
Simulation6	-0.037443	-0.033461	-0.04192	-0.16463	0.59182	-0.16545	-0.042631	-0.034999	-0.040006	-0.031283
Simulation7	-0.035843	-0.030094	-0.033044	-0.041776	-0.16545	0.59139	-0.16651	-0.044574	-0.044733	-0.029379
Simulation8	-0.035065	-0.028628	-0.030096	-0.033355	-0.042631	-0.16651	0.58981	-0.16805	-0.056007	-0.028471
Simulation9	-0.035367	-0.028472	-0.029382	-0.031187	-0.034999	-0.044574	-0.16805	0.58611	-0.18451	-0.028574
Simulation10	-0.043801	-0.034986	-0.035767	-0.037247	-0.040006	-0.044733	-0.056007	-0.18451	0.51234	-0.035285
Simulation2	-0.18436	-0.16901	-0.044577	-0.034898	-0.031283	-0.029379	-0.028471	-0.028574	-0.035285	0.50584

Fig. 7 - Capacitance and Conductance matrices for sensing regions

- Typical values obtained for scenario 2 are:

Analysis Setup 1

Capacitance Matrix [pF]

	Simulation1	Simulation2	Simulation3	Simulation4	Simulation5	Simulation6	Simulation7	Simulation8	Simulation9	Simulation10
Simulation1	0.25899	-0.088657	-0.024987	-0.019203	-0.016592	-0.015356	-0.014499	-0.01402	-0.013995	-0.017221
Simulation2	-0.088657	0.29306	-0.081661	-0.019964	-0.014875	-0.013063	-0.012045	-0.011509	-0.011415	-0.013993
Simulation3	-0.024987	-0.081661	0.29551	-0.080618	-0.018889	-0.014313	-0.012533	-0.011703	-0.011475	-0.013977
Simulation4	-0.019203	-0.019964	-0.080618	0.29431	-0.077958	-0.018721	-0.014121	-0.012518	-0.011995	-0.014437
Simulation5	-0.016592	-0.014875	-0.018889	-0.077958	0.29115	-0.077788	-0.018538	-0.014155	-0.012881	-0.015142
Simulation6	-0.015356	-0.013063	-0.014313	-0.018721	-0.077788	0.29427	-0.079806	-0.019025	-0.014936	-0.016659
Simulation7	-0.014499	-0.012045	-0.012533	-0.014121	-0.018538	-0.079806	0.2956	-0.080169	-0.018863	-0.019105
Simulation8	-0.01402	-0.011509	-0.011703	-0.012518	-0.014155	-0.019025	-0.080169	0.29484	-0.081263	-0.024882
Simulation9	-0.013995	-0.011415	-0.011475	-0.011995	-0.012881	-0.014936	-0.018863	-0.081263	0.29336	-0.088553
Simulation10	-0.017221	-0.013993	-0.013977	-0.014437	-0.015142	-0.016659	-0.019105	-0.024882	-0.088553	0.28801

Conductance Matrix [mSie]

	Simulation1	Simulation3	Simulation4	Simulation5	Simulation6	Simulation7	Simulation8	Simulation9	Simulation10	Simulation2
Simulation1	0.51274	-0.056074	-0.044818	-0.039971	-0.037443	-0.035843	-0.035065	-0.035367	-0.043801	-0.18436
Simulation3	-0.056074	0.50972	-0.16647	-0.042516	-0.033461	-0.030094	-0.028628	-0.028472	-0.034986	-0.16901
Simulation4	-0.044818	-0.16647	0.59101	-0.16493	-0.04192	-0.033044	-0.030096	-0.029382	-0.035767	-0.044577
Simulation5	-0.039971	-0.042516	-0.16493	0.59051	-0.16463	-0.041776	-0.033355	-0.031187	-0.037247	-0.034898
Simulation6	-0.037443	-0.033461	-0.04192	-0.16463	0.59182	-0.16545	-0.042631	-0.034999	-0.040006	-0.031283
Simulation7	-0.035843	-0.030094	-0.033044	-0.041776	-0.16545	0.59139	-0.16651	-0.044574	-0.044733	-0.029379
Simulation8	-0.035065	-0.028628	-0.030096	-0.033355	-0.042631	-0.16651	0.58981	-0.16805	-0.056007	-0.028471
Simulation9	-0.035367	-0.028472	-0.029382	-0.031187	-0.034999	-0.044574	-0.16805	0.58611	-0.18451	-0.028574
Simulation10	-0.043801	-0.034986	-0.035767	-0.037247	-0.040006	-0.044733	-0.056007	-0.18451	0.51234	-0.035285
Simulation2	-0.18436	-0.16901	-0.044577	-0.034898	-0.031283	-0.029379	-0.028471	-0.028574	-0.035285	0.50584

Fig. 6 - Capacitance and Conductance matrices for stimulation regions

The following are typical outputs useful for studying field distributions in terms of electric field and current density: both images refers to an input voltage of 1V.

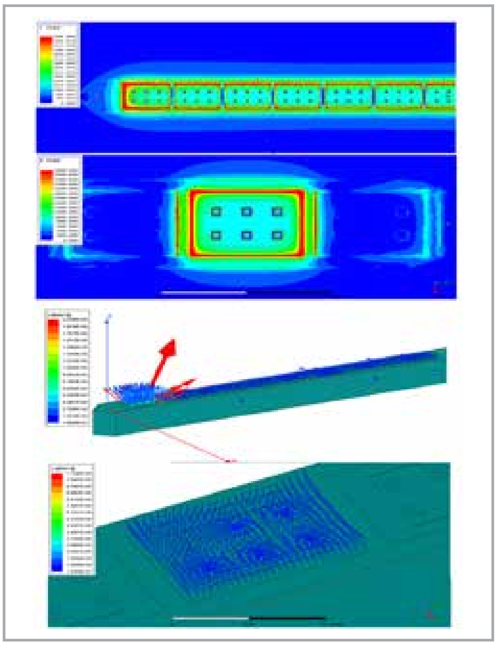


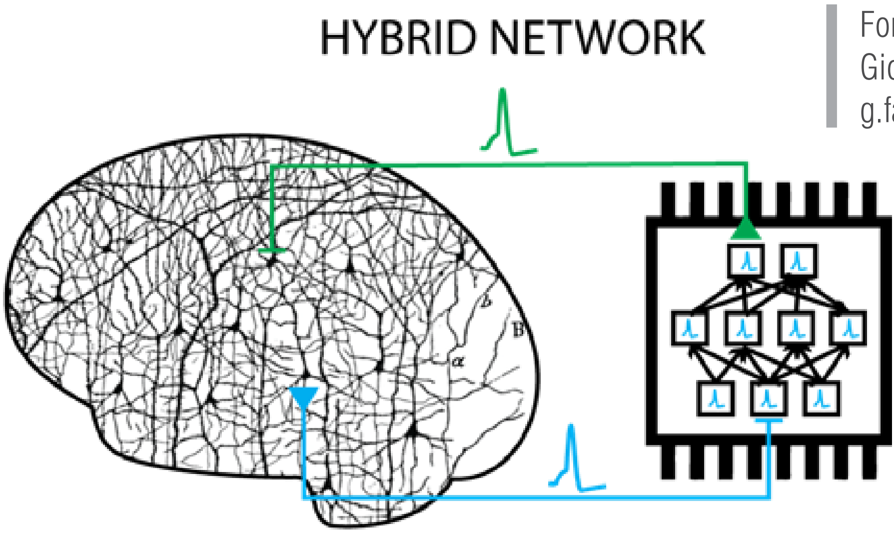
Fig. 8 - Contour graphs of the electric field over the microelectrodes (top); Vector graphs of the current density within the surrounding conductive dielectric with properties similar to neural tissue (bottom).

Conclusion

The SYNCH project started in 2019 and continues to the end of 2022. Significant progress has been made in this first year of activity.

In this article we have described the method generated by EnginSoft to design new classes of microelectrodes for neuroimplants. EnginSoft will investigate further developments in "System Simulation".

For further information on the SYNCH project, visit: https://synch.eucoord2020.com/



For more information:
Giovanni Falcitelli - EnginSoft
g.falcitelli@enginsoft.com

The Grottaglie Airport Test Bed

The need for an Experimentation Facility

By Roberto Guida and Antonio Zilli
DTA

The economics of the unmanned aerial vehicle (UAV), or drone, industry today means that players must seek new business growth opportunities and smart ways to win new markets. New technologies to develop innovative products and services for drone users are essential.

Remotely piloted aircrafts can be usefully applied in a wide variety of sectors: from land observation to enhance soil productivity (i.e. smart agriculture), to monitoring of air quality and territories to prevent and mitigate damage from natural risks and disasters such as earthquakes, avalanches, landslides, etc., to the inspection of urban assets and the patrolling of national borders. In the past weeks, drones were also widely used in the COVID-19 emergency

Fig. 1 - Service packs for different drone categories (© European Union, [2020] Responsibility for the information and views in this article lies entirely with the author(s)).

management framework. Lastly, UAVs are expected to be used in UAM operations that means to transport goods or passengers in wide metropolitan areas.

Increases in the flight range of unmanned vehicles, developments in Internet of Things (IoT) technologies, the miniaturization of satellite navigation (SATNAV) devices, as well as other innovations, enable more effective, efficient and safe use of UAVs. These new capabilities and applications have generated the need to (among others):

- define UAV flight rules;
- design and develop new technologies, applications and systems to govern the use of UAVs in unsegregated air space and in very low-level air space;
- implement, test and certify platforms;
- train the various operators involved (pilots, payload operators, air traffic controllers, etc.);
- develop applications to process the data collected through the UAV payload sensors.

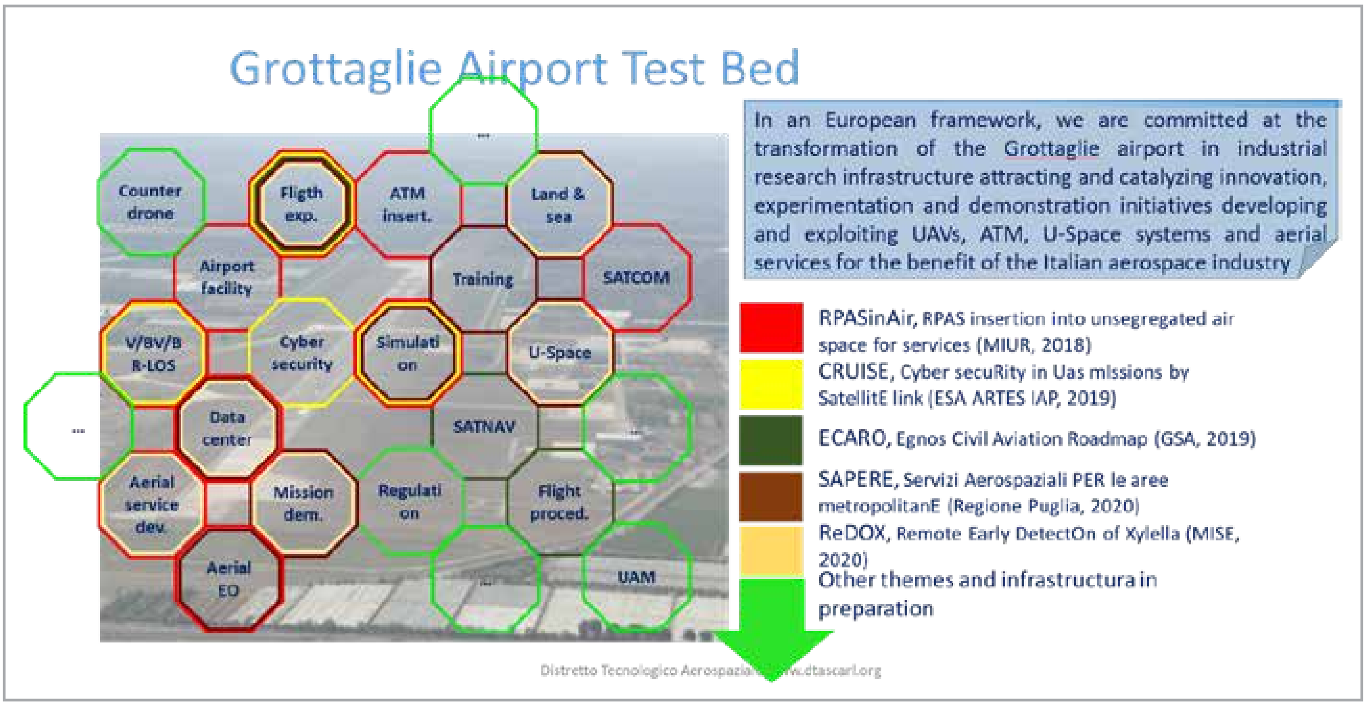


Fig. 2 - Roadmap for the development of the Grottaglie airport Testbed

While there are several European entities (the European Air Safety Agency – EASA, the Single European Sky ATM (air traffic management) Research Joint Undertaking – SESAR JU, the European Organization for the Safety of Air Navigation – Eurocontrol, etc.) that are highly committed to the development of the UAV industry, the aerospace industry itself is also engaged in a number of national and international R&D projects with the ultimate objective of demonstrating the feasibility of and the capacity to remotely pilot an aircraft system in complex aerial scenarios. This complexity is defined by the presence of other aerial vehicles or the presence of buildings (urban scenarios).

The need for a facility able to provide the resources necessary to conduct such a wide range of testing, demonstration and training services is therefore crucial: not only to support the development of the systems and applications themselves, but also to establish and manage preferential channels of communication with the aviation authorities in order to facilitate the regulatory development process.

Italy has identified an entire airport that is both equipped and operational into which this infrastructure can be introduced: ***the "Marcello Arlotta" airport of Taranto-Grottaglie.***

THE GROTTAGLIE AIRPORT TESTBED AND ITS PILLARS

Grottaglie Airport was awarded a mission by the Italian Ministry for Infrastructure and Transport to support the development and growth of the Italian aerospace industry. In January 2015, ENAV, the Italian air navigation service provider, created three segregated air spaces (totaling about 370km2) where drone products, operations and services can be tested and demonstrated. Since then, the Distretto Tecnologico Aerospaziale (DTA – the aerospace technological district) has devised an impressive development strategy centered around this airport infrastructure to increase regional and national capacity in UAV development, applications and services.

Numerous regional, national and European research, innovation and demonstration projects aimed at developing new knowledge, implementing innovative applications and equipping the airport facility with all the required systems for an aeronautical testbed, have been proposed and launched (in collaboration with members and partners).

Developments started from creating an ICT environment that can simulate/emulate and design real flight tests to improve the architectures and that can integrate other modules for testing new branches of UAV technology (cybersecurity, autonomous flight, etc.).

RPASinAir consortium: Distretto Tecnologico Aerospaziale (DTA), ENAV, Telespazio, Università di Bari, Vitrociset, Aeroporti di Puglia, Università Kore di Enna, Istituto Nazionale di Fisica Nucleare, Leonardo, Centro Nazionale delle Ricerche (CNR), the Italian National Agency for New Technologies, Energy and Sustainable Economic Development (ENEA), Planetek, Enginsoft, Politecnico di Bari.
CRUISE consortium: Planetek Italia, DTA, Leonardo, Aeroporti di Puglia, ENAV, Telespazio.
ECARO consortium: ENAV, DTA, Ums Skeldar, Airgreen.
SAPERE consortium: DTA, Planetek Italia, the Institute for electromagnetic sensing of the environment (CNR IREA), Università degli Studi di Bari, Leonardo
REDOX consortium: DTA, CNR, ENAV, in collaboration with Planetek

	RPASinAir – Integration of remotely piloted aircrafts in non-segregated airspace RPASinAir develops technologies and solutions that drive the integration of remotely piloted aircraft systems (RPAS), especially beyond radio line of sight (BRLOS) operations, in non-segregated airspace Expected results are: 1. An infrastructure and a laboratory for the simulation of BRLOS operation of RPAS 2. SATCOM and SATNAV aviation receiver requirements, 3. Innovative air traffic control (ATC) solutions and applications 4. payload sensors and data processing systems and applications The ultimate goal is to design and enable innovative monitoring services (risk prevention, emergency management, etc.). Aerial services will be demonstrated through real flights operated from Grottaglie.
	CRUISE – Cyber secuRity in Uas mIssions by SatellitE link The project CRUISE (January 2019, June 2022) addresses the European requirement for higher cyber security of RPAS. Project partners will design, develop and validate a CyberSec Test Range to provide cyber vulnerability and resiliency assessment services for RPAS to a wide EU community of research and industrial users. The CyberSec Test Range will establish a combination of services combining ICT systems and SATCOM and SATNAV services to evaluate the overall vulnerability of UAS, specifically the analysis of flight platforms (including the ground pilot station), on-board avionics and payload sensors. Quality, consistency and integrity of data collected during visual line of sight (VLOS), radio line of sight (RLOS), and BRLOS operations will be verified as well.
	ECARO – Egnos Civil Aviation Roadmap ECARO is a project, funded by GSA (European Global Navigation Satellite Systems Agency) and coordinated by ENAV in partnership with DTA, Planetek, UMS Skeldar and Airgreen, to develop products based on the European Global navigation Satellite System (GNSS) – the European Geostationary Navigation Overlay Service (EGNOS) and Galileo, and contribute significantly to their use in Europe. ECARO's main objectives include: • Design, validation and publication of GNSS procedures (ILS Cat.I-like) based on EGNOS in four airports: Bari-Palese, Genova-Sestri, Torino-Caselle, Trieste • Development of a mobile validation platform for GNSS procedures • Design, validation and publication of three Low Level Routes (Tirrenica, Adriatica, Appenninica) for rotorcraft operations • Design and demo execution of GNSS procedures for RPAS for Grottaglie airport and assessment of threats
	Aerospace services for metropolitan areas Definition and development of innovative solutions based on the exploitation of images and data acquired from satellites or remotely piloted aircrafts airborne sensors aimed at territory management. The simulation laboratory deployed during RPASInAir will be employed for the definition of CONOPS of the available technologies and for the planning of EO missions.
	Remote Early DetectOn of Xylella Definition, design, develop and prototype a process and the related systems and applications to detect olive trees infected by Xylella. Detection applications will collect massive images from drone flights and will trigger terrestrial monitoring. By this way Xylella will be detected much earlier its visible impact appears.

Fig. 3 - Segregable air spaces

A key milestone in the construction of the test center is the (ongoing) deploymentof a hyperconvergent infrastructure to simulate and emulate missions and/or components, as required by the objectives of the RPASInAir project. This first part of a broader architecture will allow the testing of Beyond Radio Line Of Sight (BRLOS) operations in Europe with a large Optionally-Piloted Vehicle (OPV), the Leonardo SW4-SOLO.

The pillars upon which the Grottaglie Airport Test Bed is being created are summarised in Fig. 2, which presents a roadmap for the development of a service bundle (a description of the projects can be found in the table). In order for these projects to achieve their desired results, it is important to consolidate a collaborative network with the partners.

Involving other companies and authorities – such as the Italian civil aviation authority (ENAC), ENAV, Telespazio, the European Space Agency (ESA), the Italian space agency (ASI), Aeroporti di Puglia, the Italian airforce, and the Italian aerospace research agency (CIRA) – that have common aims and research goals, which can operate synergistically within the Grottaglie Airport Test Bed enables the creation of a cradle for industrial development at the Grottaglie Airport.

For more information:
Roberto Guida, Antonio Zilli - DTA
roberto.guida@dtascarl.it
antonio.zilli@dtascarl.it

Composite lightweight redesign of EV closures in the QUIET Horizon 2020 project

eCon Engineering uses simulation to design and optimize a new carbon composite structure

By Tamas Turcsan
eCon Engineering

The QUIET project (QUalifying and Implementing a user-centric designed and EfficienT electric vehicle) targets weight reduction and thermal inertia while improving the thermal insulation of an electric vehicle (EV) to improve efficiency without reducing performance. It aims to develop an improved and energy efficient electric vehicle with an increased driving range for real-world driving conditions. This is being achieved by exploiting the synergies of a technology portfolio (see Fig. 1) in the areas of:

- user centric design with enhanced passenger comfort and safety,
- lightweight materials with enhanced thermal insulation properties, and
- optimized vehicle energy management

eCon Engineering, as the work package leader for lightweight materials with enhanced thermal properties, is working with composites and novel hybrid foam materials to reduce the weight of EV closure elements while improving their thermal properties in close cooperation with other members of the consortium e.g. Austrian Institute of Technology and Fraunhofer IFAM.

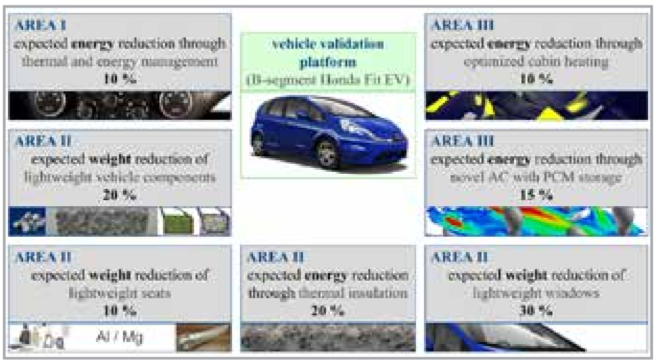


Fig. 1: Target vehicle and areas of QUIET project

The QUIET Work Package 3 (WP3) aimed to develop lightweight glazing, closure elements (e.g. side doors, trunk lid and engine hood) and seats, all with improved thermal properties. The goal was to reduce weight by 30% for glazing, 20% for closure elements, and 10% for the lightweight seat structure, in addition to improving the thermal properties in terms of the demonstrator vehicle's energy consumption during heating or cooling of the cabin.
The development process (see Fig. 2a) began with data collection and analysis of the current structure to estimate the potential for mass reduction and for optimization of the thermal properties. From the results for the original structures (which served as the baseline), an extensive research and development process was conducted using material sciences, computer aided design and the finite element method (FEM). In each case, a multi-step iteration process was performed to optimize the newly developed structures and to achieve the best possible result. At the end of the development, the previously

About eCon Engineering Ltd

eCon Engineering Ltd was founded in 2002 in Hungary and currently employs 70 engineers. With almost 20 years of experience in FEM, CFD, BEM and automated machine building competencies, the company solves a variety of problems in the automotive, aerospace, energy, heavy duty, bus, composite, and railway industries.
The 25 engineers in our CAE Services department provide tailored solutions from meshing and model building, through collaboration with in-house CAE teams, to complete simulation of development processes, FEA, CFD, MBS and 1D simulations, with a wide range of CAE software, including Ansys, Moldex3D and Cast-Designer.

set goals for weight reduction as well as the thermal properties were achieved.

For the lightweight closure elements, a carbon composite design was developed and optimized according to a predefined development process, using weight-cost and weight-strength trade-offs. To do this, a design of experiments (DoE) process was used to identify areas where composite redesign could be effective and areas where preserving the original structure would be more useful. In this work phase, simulations were performed using Ansys Composite PrepPost (ACP) to construct models under static loads, and LS-DYNA to calculate the side collisions (see Fig. 2b). At the end of the development process, trade-offs between mass, cost, thermal properties, and stiffness were ascertained, and the best possible composite-layer structure for each target component was determined. Not only were the lightweight structures designed, but they were also manufactured as prototypes (see Fig. 3) to be implemented in each case in the demonstrator vehicle of the QUIET project. Therefore, the result of this work package was not only a new design concept and production plan, but also real parts with significant weight reduction and better thermal insulation or lower heat capacity, which can be implemented and tested in the last phase of the project, in 2020.

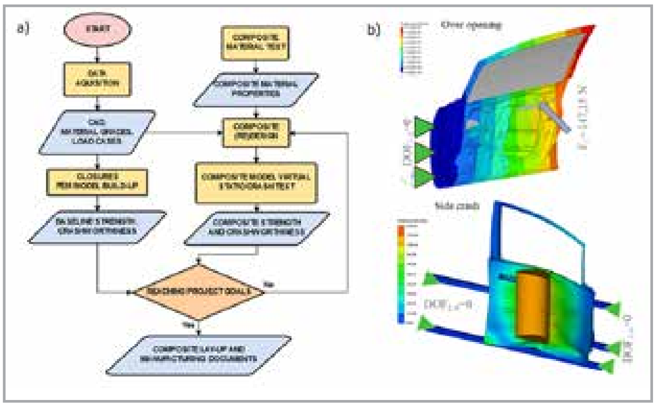


Fig. 2 - Development process (a) and virtual testing of the side door with FEM (b)

Fig. 3 - A manufactured prototype of the composite side doors of QUIET project

In all sub-tasks of WP3, we also calculated the possibilities of economic upscale. They show what the costs of using newly developed solutions would be not only for prototype vehicles but also for higher series vehicles.

Acknowledgement:
This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No. 769826. The content of this publication is the sole responsibility of the Consortium partners listed herein and does not necessarily represent the view of the European Commission or its services.

For more information:
Tamas Turcsan - eCon Engineering
tamas.turcsan@econengineering.com

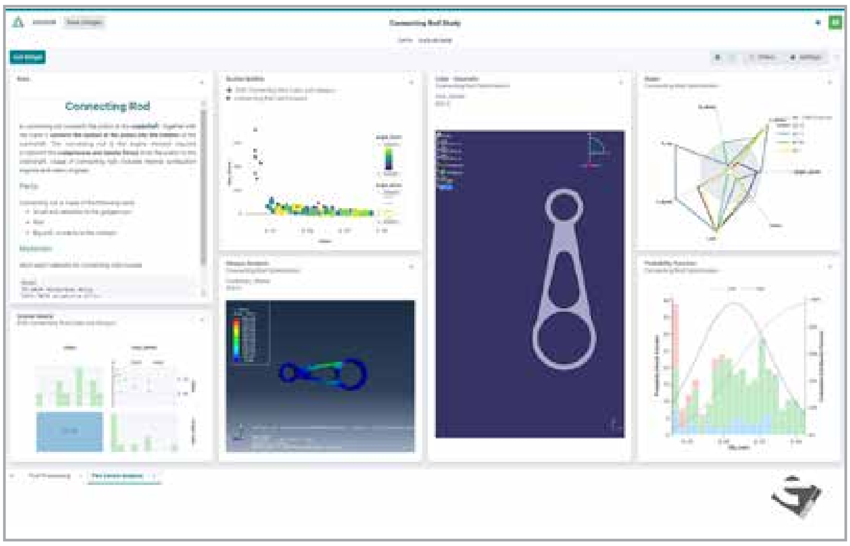
EnginSoft Turkey celebrates its fifth anniversary

EnginSoft Turkey was established in Istanbul Turkey at the Teknopark İstanbul and Teknopark Bulvarı in 2015. Since then, the branch has grown its operations in the area in many different sectors, including advanced engineering services, on-site engineering, software sales such as Ansys, modeFRONTIER, ParticleWorks, CETOL 6Sigma and MapleSoft, offset and R&D projects.

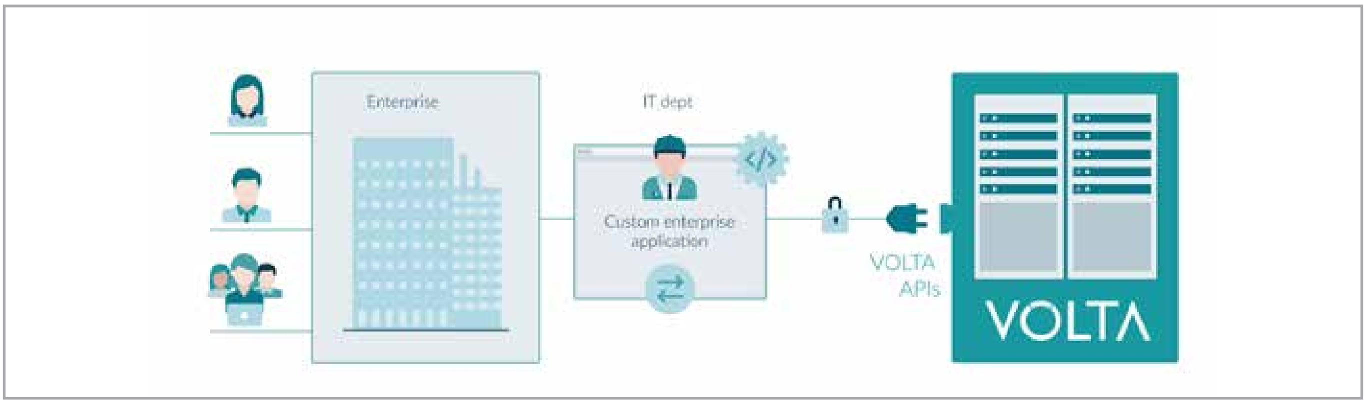
ESTECO announces the availability of the 2020 Spring Release of VOLTA and modeFRONTIER

Democratize the simulation design process and accelerate product development using smart data analysis, API-fication and fully-autonomous design exploration

Companies want to use simulation to shorten product development time, however, they commonly rely on just a few experts to manage the large amount of simulation data that needs to be compared and validated by multiple departments. This often leads to an unnecessarily complicated and time-consuming analysis process. Collaboration and democratization are fundamental to empower more engineers and to take full advantage of simulation technology. VOLTA, ESTECO collaborative web platform for simulation process and data management (SPDM) and design optimization, is designed specifically for this purpose: it makes every simulation process a collaborative, enterprise-wide experience to deliver better products, faster.



VOLTA advisor new visualization capabilities



External systems can communicate with the VOLTA APIs

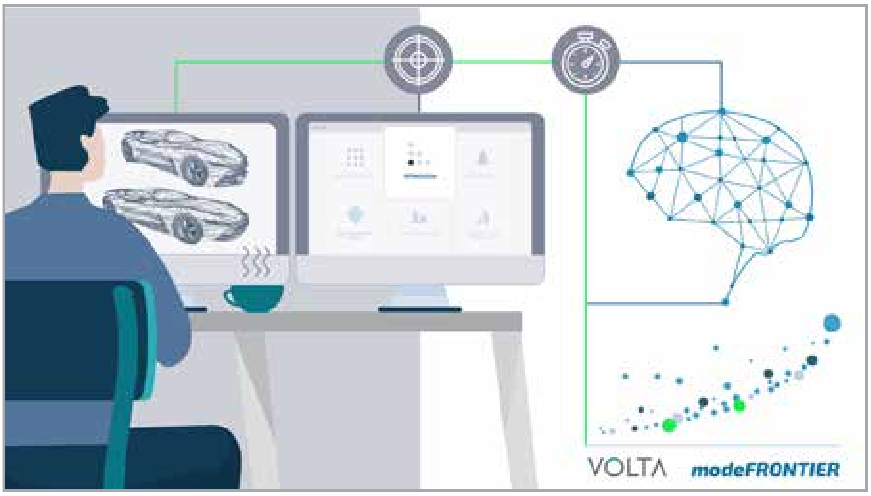
Through its web applications, the VOLTA platform is continuously evolving to expand the use of engineering simulation across teams and to offer an ever easier-to-use optimization experience. In the new 2020 Spring Release, VOLTA advisor, the advanced data analysis app, includes a smart visualization dashboard that allows users to take notes and add images to make data easier to read. This streamlines the comparison and evaluation of different design solutions and supports analysis with information to aid easy understanding of the simulation results.

Another step towards democratization in the simulation process has been achieved with the new VOLTA application programming interfaces (APIs) that have been enhanced in the latest release. Companies can now unleash the full power of enterprise data by securely connecting their different apps, platforms, services, and systems with the VOLTA platform. This enables non-simulation experts, suppliers, and even clients to interact with the engineering data they need, instead of getting lost in the complexity of the whole simulation process. In fact, different users can directly interact with VOLTA through a simplified web interface. Based on their permission level, they can request and run simulations, apply optimization strategies or just download results. The new VOLTA APIs deliver a superior digital experience for the engineering design process: enhancing collaboration across teams, increasing productivity, and reducing operational costs while maintaining appropriate security controls.

The 2020 Spring Release focuses on increasing the quality in a simulation while keeping the process simple. Autonomous design-space exploration is now available both in VOLTA enterprise platform and in modeFRONTIER desktop solution for process automation and optimization. Users can rely on the intelligent algorithms without needing to make any settings whenever they approach an engineering problem or have to optimize their designs. This means they can take advantage of the automatic approach to machine learning and focus on the design improvements rather than losing time in configuring the optimization strategies. ESTECO highly trusted autonomous Artificial Intelligence technique enables its users with the best trade-off between the quality of the solution and the time necessary to find it.

During simulation, engineers need to consider uncertainties in the early design stage to avoid possible manufacturing errors during the production phase. In addition to ESTECO consolidated and robust design optimization technology to predict product performance, users can now also rely on the Taguchi-based quality engineering method. This becomes very useful when it is more cost effective to minimize the impact of product/process failures (noise factors) than to control the causes of variations, especially in the context of component manufacturing tolerances. The new modeFRONTIER quality engineering module allows R&D engineers to simulate noise and identify the best set of parameters to make the entire system less sensitive to uncertainty. This innovative approach requires very few design evaluations and a short computational time to reach the perfect balance between product and engineering quality and manufacturing costs.

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



The autonomous approach to design-space exploration available in VOLTA and modeFRONTIER

“to take care with care”

New PREMUROSA project will improve the treatment of musculoskeletal disorders

A group of international partners from Italy, Finland, Portugal, Ireland, Latvia, Serbia and Switzerland launched an EU-funded project called “Precision medicine for musculoskeletal regeneration, prosthetics, and active ageing” (PREMUROSA). The three-year programme’s main objective is to train a new generation of tissue engineering scientists from multiple European countries to develop new technologies and therapies for musculoskeletal disorders.

Musculoskeletal diseases reportedly affect roughly half of those over 60, strongly impacting on their quality of life and placing a significant burden on healthcare and welfare systems. Treatment of musculoskeletal disorders is currently either based on prosthetic or regenerative surgical procedures, often involving the implantation of medical devices compromising the effectiveness of treatments.

A significant improvement could be achieved by using precision medicine, specifically designed for the patient’s individual characteristics. This requires new and highly skilled professionals to develop new strategies to translate tissue engineering innovations into useful information for customising therapies, taking into account the characteristics of each patient.

The project aims to train a new generation of scientists with an integrated vision of the entire value chain of musculoskeletal regeneration technologies that will increase the innovations needed to achieve principles of precision in the development of innovative devices and optimized clinical applications.

Thirteen students from the fields of medical and health sciences, clinical medicine, surgery and surgical procedure will be selected to participate in the interdisciplinary project and earn their PhD degrees. They will be hosted in turn by a member of a European consortium consisting of universities, research institutions and companies from Italy, Switzerland, Portugal, Finland, Latvia, Ireland and Serbia.

These young researchers will acquire advanced knowledge and skills through an innovative combination of academic, industrial and clinical experience and training. They will gain the benefit of an excellent scientific environment, up-to-date technology and supervision by international industry leaders.

The successful implementation of PREMUROSA will not only lay the foundation for innovative PhD training, but will also contribute to addressing important social challenges, such as optimizing clinical choices and thus improving patients’ quality of life, while reducing the costs of the healthcare system. In addition, industrial competitiveness will be significantly enhanced through the optimization of medical devices and the development of new products in the project.

The consortium is comprised of eleven European partner institutions, led by the University of Eastern Piedmont (Italy), and several non-academic partners and companies specialising in the biomedical field. Also mong the eleven partner organisations are: the Rizzoli Orthopaedic Institute (Italy), Aalto University Foundation (Finland), AO Research Institute Davos (Switzerland), Riga Technical University (Latvia), the Faculty of Technology and Metallurgy, University of Belgrade (Serbia), the National Institute of Biomedical Engineering (INEB), University of Porto (Portugal), the Polytechnic University of Turin (Italy), Tampere University of Technology, (Finland), the National University of Ireland (Ireland), and EnginSoft (Trento, Italy).

Carla Baldasso, international research officer at EnginSoft states, “EnginSoft will brings its industrial experience to the biomedical field. As a leader in optimization techniques, data modeling and simulation, the EnginSoft Industrial Applied Mathematics Group is well placed to contribute to the project. In-vitro and in-silicone models, data and results will be housed in a decision support systems (DSS) designed to support physicians in identifying the most appropriate patient-centred medical treatment. In addition, EnginSoft will bring to the project its experience in the exploitation of project results.

About Premurosa

PREMUROSA is part of the Marie Sklodowska-Curie Innovative Training Network – a European Joint Doctorates programme, funded by the European research and innovation programme, Horizon 2020. The project aims to train a new generation of creative, entrepreneurial and innovative early-stage researchers, able to face current and future challenges and to convert knowledge and ideas into products and services for economic and social benefit.

For more information:
Carla Baldasso - EnginSoft
c.baldasso@enginsoft.com

Ansys 2020 R1 features new fluid dynamics tools for all engineers

Improvements to ease of use and efficiency for various CFD applications

With Ansys 2020 R1 fluid dynamics, all engineers with different level of experience can now streamline their product development life cycles. The automated workflows allow all members of an engineering team to setup all the physics more quickly and easily. The following article lists the main developments, new features and improvements that have been introduced for the different computational fluid dynamics (CFD) tools.

Ansys pre-processing with Fluent Meshing

Ansys Fluent Meshing has been enhanced to organize the mesh generation process with a task-based workflow. The user is now guided step by step through a sequence of tasks, providing a streamlined end-to-end wizard to reduce the time to the solution. This approach eliminates the preceding entry-level barrier, offers the power of Fluent Meshing to new users and allows deployable custom workflows for improved automation.

Two different workflows are available: Watertight Geometry Workflow (for clean geometries) and Fault Tolerant Meshing Workflow (for problematic and complex geometries). In the recent releases, Fluent Meshing has undergone significant development and, in 2020R1, the two guided workflows have been enhanced in term of scalability, usability and flexibility.

Local boundary layers can now be applied to more specific portions of the model, such as only the walls, just the fluid regions, just the solid regions, the fluid-solid interfaces, named regions, or to specific zones or labels. It is therefore possible to differentiate the prism settings on different surfaces to accurately capture the boundary layer and the geometry details.

Parallel grid generation is now available for the Tet-Hexcore method, while it has been improved for the Poly-Hexacore method, specifically with regard to scalability and quality of high aspect ratio poly prisms.

The ***Watertight Geometry Workflow*** makes it possible to create linear arrays of parts, which is particularly useful for linear periodic geometries such as batteries. For quality diagnostics, the workflow can automatically draw bad volume cells to enable better understanding of mesh issues and problems. The connection with Workbench has been improved to have a robust, fully automatic procedure to connect Space Claim, Fluent Meshing, Fluent Solver and all the other tools of this platform.

The ***Fault Tolerant Workflow*** allows the Identify Deviated Faces task to be used to see how the mesh of the wrapped surface matches the original geometry. This is especially useful for monitoring the wrapping process and evaluating its quality and fidelity. There is a new type of Offset Box in the Create Local Refinement Regions task that allows the creation of a scaled bounding box to easily identify and modify the regions where a finer mesh is required.

A new task has been introduced to better manage the setting of boundary conditions: with the Update Boundaries task, the user can easily identify the different surfaces and apply the correct boundary conditions, which are transmitted directly to the solver.

Ansys FLUENT

The last release of Ansys Fluent introduces new capabilities and overall improvement at different levels.

Fluent User Interface

A New Fluent Launcher has been added, which provides a ***single launch point*** for both general purpose Fluent and application work packages (Meshing, Solution, Icing). After selecting a file to start with, the dimensions (2D or 3D) and precision (double or single) are automatically updated for the case. New effects such as reflections and shadows are included and enabled by default if suitable graphics hardware is found.

Common settings for boundary conditions can be configured directly in the ***quick-edit boxes*** that appear when you select a boundary in the graphics window.

New shortcuts can quickly change the displayed colors of boundaries with calculated variables for quick inspection of

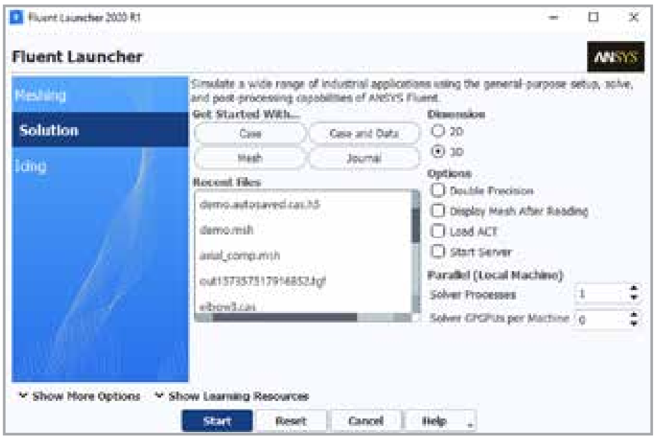


Fig. 1 - New Fluent Launcher

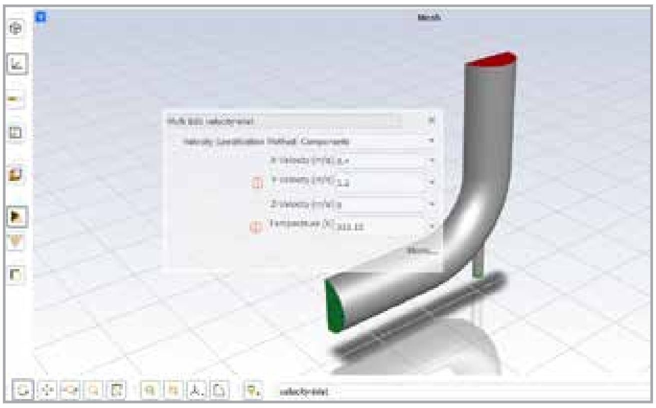


Fig. 2 - Quick Edit box for boundary conditions

some results. The interactive plane tool has been completely re-designed for more intuitive handles. Case and Data files are now written in ***h5 (cff) format by default***, which offers better performance, especially for large cases.

Physics Modeling

The biggest change for Turbulence is a modification of the default selected model: now the kw-SST is automatically activated for any new case. For Eulerian Multiphase flows, the new release consolidates the use of the workflow into a single dialog box, where the setup moves from left to right through the tabs for model

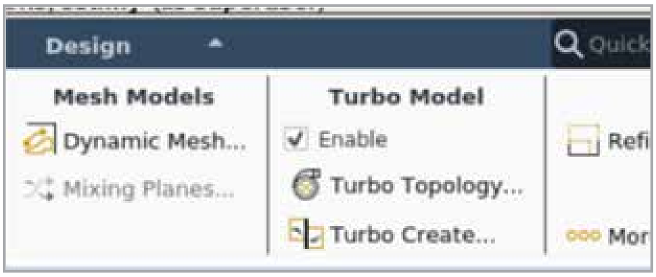


Fig. 3 - New panel for turbomachinery setup

selection, phase definition, and phase interaction. The Population balance model is now integrated into the workflow as the final tab, which eliminates the need to load the specific add-on module.

2020R1 also introduces multiphase regime transition model based on the ***Algebraic Interfacial Area Density*** (AIAD) approach, which allows more accurate simulation through the incorporation of the sub-grid turbulence contribution and mass transfer mechanisms to account for entrainment and absorption; it can predict droplet size distribution when used with population balance.

There are various enhancements to robustness for multiphase algorithm, including buoyancy-driven flows, mixtures and VOF simulations (Volume of Fluid). A simplified ***semi-mechanistic boiling model*** has been introduced under homogeneous assumptions, which is particularly useful for automotive applications operating at low pressure.

A ***non-adiabatic FGM*** (Flamelet-Generated Manifold) with premixed flamelets expands the capabilities for combustion scenarios, where species formation is significantly affected by heat loss/gain from surrounding walls and participating media. For battery simulations, a fully 3D lithium-ion (Li-ion) electrochemical model has been introduced; when the cell is discharged, the migration of Li+ ions from the anode to the cathode can be evaluated with the resulting changes in local Lithium concentration.

A new ***specialized mesh interface for Turbomachinery*** (General Turbo Interface, GTI) supports common blade row models, including frozen-rotor with pitch change, mixing plane methods and transient with pitch change.

Solver Enhancements
Automatic One-to-One Interface pairing has been improved for configuring non-conformal interfaces for multiple connected bodies (complex CHT cases); this new option will create a unique interface definition for each combination of contacting bodies, even if the faces have not been decomposed during meshing. Conformal and non-conformal periodics are now configured in a single GUI with an automatic algorithm. The overset mesh in parallel has been significantly improved with better efficiency.

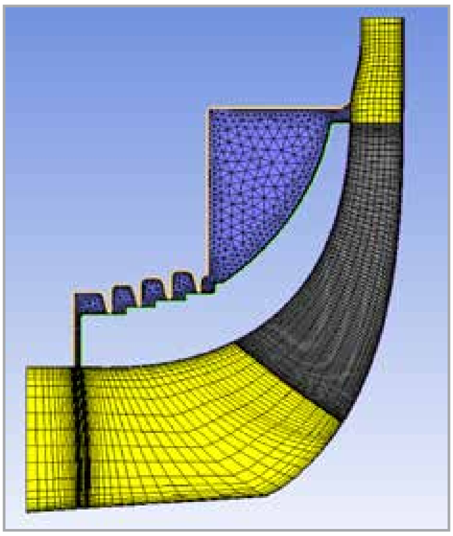


Fig. 4 - Hybrid mesh of Ansys TurboGrid

Ansys CFX and Turbo Tools

Ansys is constantly developing new capabilities to help customers conduct accurate turbomachinery simulations. The main improvements in release 2020 R1 include the following:

Ansys BladeModeler is integrated with Ansys DesignModeler to deliver complete 3D geometry modeling capabilities and allow any number of geometric features, such as hub metal, blade fillets, cut-offs, and trims to be added. It is now possible to specify variable blade blends between leading edge and trailing edge.

Ansys TurboGrid software includes novel technology that targets complete automation combined with an unprecedented level of mesh quality for even the most complex blade shapes. Hybrid meshing can now be used to create conformal mesh for blade tips to account for motion due to flutter and for secondary flow paths and axisymmetric cavities. In 2020 R1 the main flow path can now be defined using profiles or coordinate points while CAD curves are used to define the secondary flow path. It is also possible to create separate high and low blade regions (pressure/suction surfaces) directly in TurboGrid.

Ansys CFX is a high-performance CFD software tool that delivers reliable and accurate solutions quickly and robustly across a wide range of CFD and Multiphysics applications. CFX is recognized for its outstanding accuracy, robustness and speed when simulating turbomachinery, such as pumps, fans, compressors and gas and hydraulic turbines.

Harmonic analysis speeds solutions to multistage, transient blade row problems, by calculating as few as one blade per row and reducing computational time by a factor of 100 (versus a full wheel solution) and a factor of 25 (versus the previous state-of-the-art, Fourier transform method). In 2020 R1 the solution method provides 2X faster convergence and larger stability limits and it expands application to simulating fans under inlet/outlet disturbances and modeling asymmetric flows in radial compressors or turbines. This is possible thanks to the new multifrequency analysis.

Operating Maps were first released in 19.2 and provide a streamlined workflow to run multiple operating points and build the operating map for a machine. A single simulation is set up in CFX-Pre, with ranges defined for the boundary conditions – for example, a pressure range and a rotation speed range could be defined via a CEL Table function. A single job is then submitted which spawns multiple solver runs to cover the defined operating range. The set of runs is managed and monitored as a single job and it can be post-processed in a single session. In 2020 R1 Ansys continues to build on the capabilities released in 19.2 by allowing concurrent solves – so multiple operating points can be solved simultaneously. Job monitoring and post-processing were improved to provide an option to retain only residual and monitor data in the results file, which minimizes disk space usage.

The ***Blade Film Cooling*** model provides an efficient and practical way to model an array of cooling holes and the injected film cooling flow on turbine blades. The model does not require the mesh to resolve the holes and is fully compatible with periodic and moving boundaries. Recently a cylindrical hole option was added where the drilling direction is defined, resulting in an elliptical shaped hole at the blade surface. Local flow direction angles can now also be specified as part of the injected flow quantities. Improvements in 2020 R1 focus on the usability of the Blade Film Cooling model in setup, solution, and post-processing, with better visualization of injection regions in both CFX-Pre and CFD-Post.

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

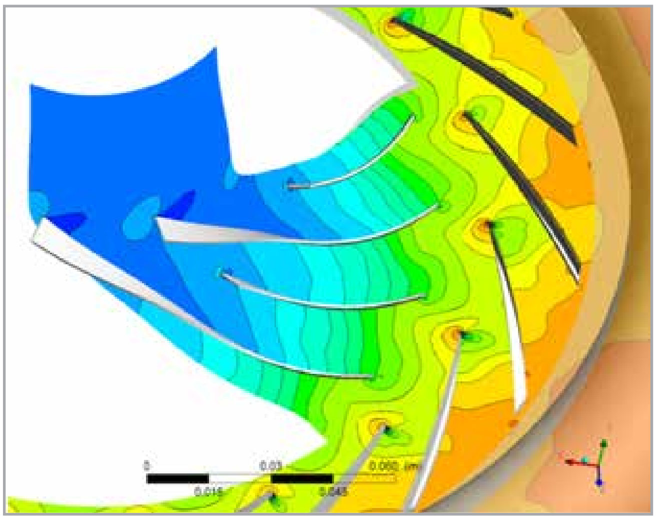


Fig. 5 - Multi-frequency Harmonic Analysis solution of a vaned impeller

Ansys Discovery Live and Discovery AIM feature new developments and improvements for design engineers

Important changes to structural and fluid dynamic capabilities

Fig. 1 - Air Velocity in a Fluid Analysis into Discovery Live

Ansys Discovery has seen lots of interesting new developments in this first part of 2020. The Ansys Discovery 2020 R1 release has many enhancements and new features, that make it a more effective tool for designer engineers. This short article will illustrate all the new features of Discovery Live and Discovery AIM, emphasising the most important from a structural and fluid dynamic perspective.

Ansys Discovery LIVE

This tool is probably the most suitable for design engineers, thanks to its ability to perform simulations in real time. Last year it already enabled us to perform structural, thermal, fluid-dynamic and dc current analyses, as well as coupled physics. Moreover, starting from a static structural analysis, it was possible to set up a topology optimization, while for all the different analyses one could configure a parameter study. All the improvements and enhancements included in the first release of 2020 are listed below.

- Improved simulation of thin parts
  The new approach of memory efficient discretization allows the simulation of thinner parts and the use of smaller GPUs
- Steady state fluids
  - Rapid solution of stead states
  - Useful for thermal/mixing problems
- Topology optimization
  - Modal support added
  - Manufacturing constraints: maximum thickness and two-sided pull direction
  - New objective: minimization of mass with stiffness and frequency (1 or both) constraints
  - Combine static and modal simulations
  - Parameter study support
- General
  - Improved rendering performance
  - Graphical display of “poorly resolved regions” highlights discrepancies between geometry and physics discretization (BETA)
  - Export to Ansys Mechanical

Improved simulation of thin parts

Probably the most significant development in Discovery Live 2020R1 is the change in memory discretization. The new approach used in this release has overcome the limitations of capturing small details, making it possible to simulate thin structures and tiny gaps. But the smartest aspect of this improvement is the efficiency of memory discretization. The new approach used in this release better allocates memory, allowing the user to capture more details while saving the GPU.

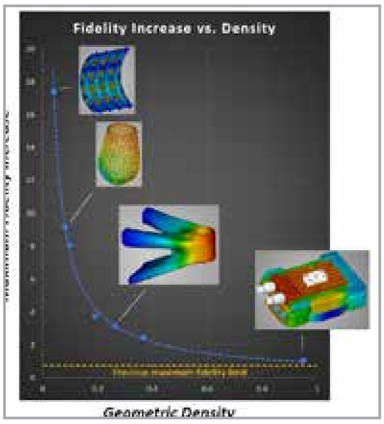


Fig. 2 - Representation of analysis fidelity in function of geometric density

Topology Optimization

Even though the Topology Optimization tool was already integrated in Discovery Live, there have been some interesting improvements made to this tool.

Firstly topology optimization studies can be based on static structural analysis, modal analysis or both. From a technical perspective this means that the target of the optimization can be reduction of volume, maximization of stiffness or the response to the first mode of vibration. In addition, new constraints, such as the pull direction or maximum thickness have been introduced.

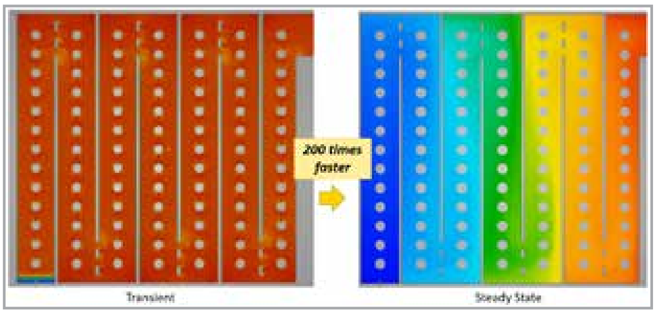


Fig. 3 – Timing of Transient analysis vs steady state analysis

Steady state fluids
Discovery Live 2020R1 enables steady-state computational fluid dynamics (CFD) analyses in addition to the transient analyses that were already available in the previous versions. This offer some advantages:

- Solutions can be found much faster than using the standard transient solver -- in many cases the solution is available after a few seconds.
- Thermal mixing problems often present large time-dependent fluctuations, due to natural physical behaviors. The same occurs with highly turbulent flows, where vortexes detach periodically from the bodies. This has consequences for the engineering parameters that are being observed (flow rates, average pressure, maximum temperatures, forces etc.).

A steady-state solution is useful in all of these cases, thus, when the design engineer is not interested in the fluctuations, but in the time-averaged solution that is more statistically meaningful.

Ansys Discovery AIM
This release also sees new features added to Discovery AIM.
Generally, these include:

- New structural material properties
  - Neo Hookean hyperelasticity
  - Linear orthotropic elasticity
- Physics aware meshing enhancements
  - Shell meshing enhancements
  - Auto hex meshing enhancements
  - Mesh quality metrics
- Beam and shell modeling
  - Truss and cable elements
  - Shell mesh expansion
- Contact usability
  - Identification of disconnected bodies
- Topology optimization
  - Manufacturing pull direction
- Live to AIM enhancements
  - Tabular transient boundary conditions
  - Steady state fluids

The principal features are discussed in more detail below.

New structural material properties
One of the most interesting new abilities in Discovery AIM 2020R1 is the introduction of two new material models. The first is Neo-Hookean Hyperelasticity, which enables the user to simulate rubber components, such as seals, O-rings, expansion, bellows, etc. The Neo Hookean Hyperelastic model is both the most common and the easiest to use in terms of constants: initial shear modulus and bulk modulus. It is based on the statistical thermodynamics of cross-linked polymer chains. This material model defines a stress-strain nonlinear curve that makes it possible to simulate rubber and polymer materials that undergo large elastic strains (20%-30%).
The second new material model included in the 2020R1 release is Orthotropic Elasticity for structures. This allows users to specify different values of Young's modulus in x, y and z directions or in other coordinate systems. Orthotropic Elasticity makes it possible to simulate many new materials, such as composite laminates.

Physics aware meshing enhancements
Discovery AIM is a Multiphysics simulation tool that includes different templates by function of the analysis's physical properties. In relation to the physics involved in the simulation it is also possible to set up the automatic mesh generator. This useful tool is enhanced in every release to improve mesh quality, and minimize user interaction to achieve high quality mesh, etc. In 2020R1 the focus of the enhancement is hex meshing. Discovery AIM default algorithm for size function is Curvature, but when treating a body without a pronounced curvature, this algorithm would cause a coarse mesh. In the new release, when this kind of geometry is detected, the mesh size of the automatic generator becomes smaller. Furthermore, if the hex mesh fails, the mesher automatically switches to a tetrahedral mesh.

Beam and shell modeling
Beam modeling was included in Discovery AIM last year. Release 2020R1 takes another step forward by introducing structural truss

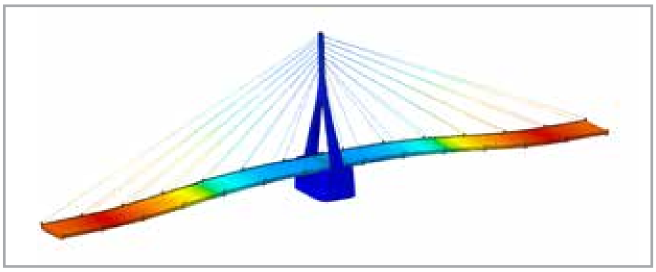


Fig. 4 – FEM model of a bridge with cable elements

and cable elements. These elements can be created as a sketch or extracted from a body using Modeling. Once a truss or cables are detected by the software, the mesh is automatically defined as a single element along the truss or the cable.
Another interesting enhancement for beam and shell modeling is the ability to expand shell meshes. This simple improvement, which is available for both physics aware mesh and the results display menu, makes it much easier to visualize the shell thickness and the shell results with a realistic representation of thickness.

For more information:
Maria Rita De Rinaldis - m.derinaldis@enginsoft.com
Alessandro Cinciripini - a.cinciripini@enginsoft.com