ENGINSOFT

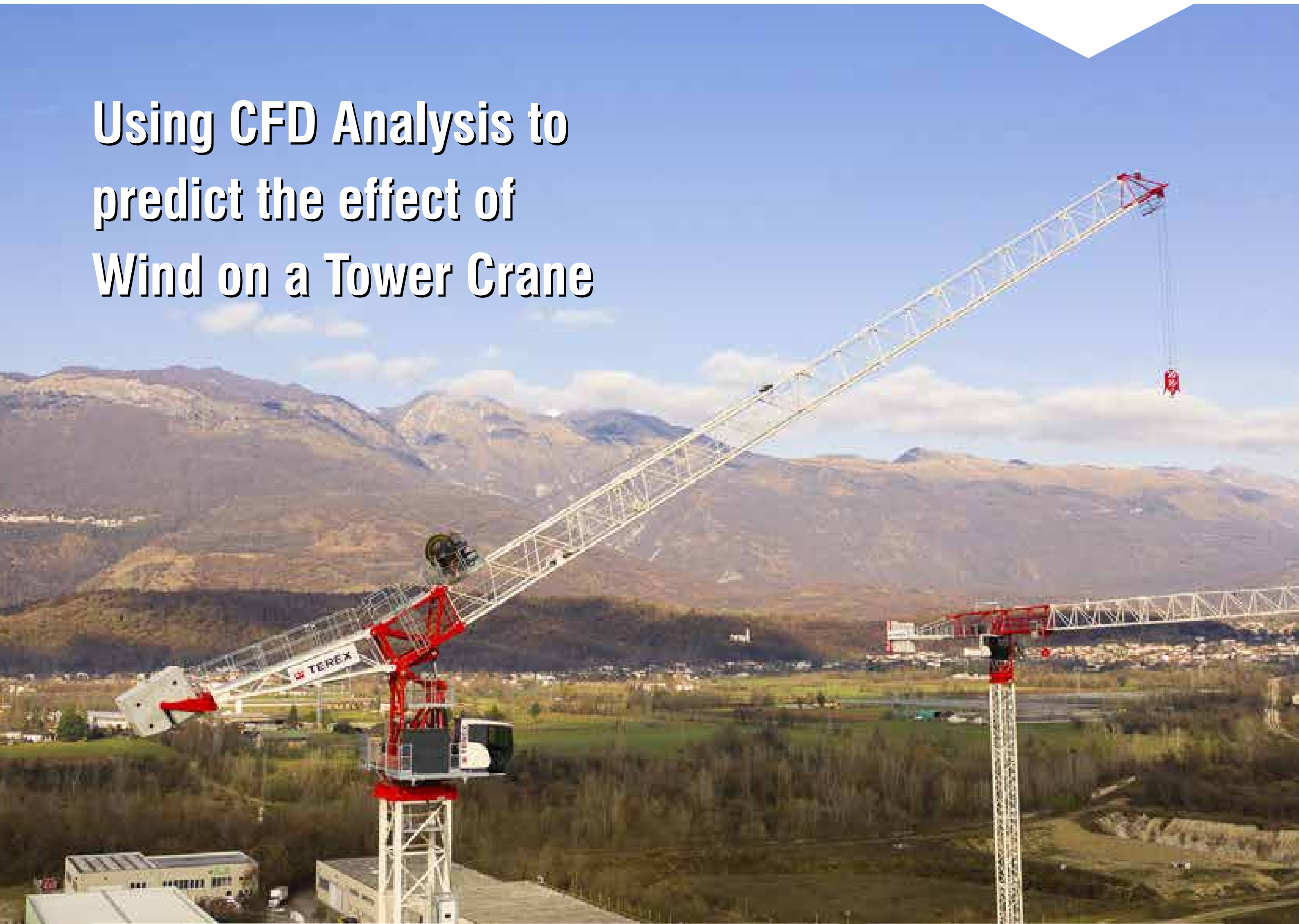
The Simulation Based Engineering & Sciences Magazine

Newsletter

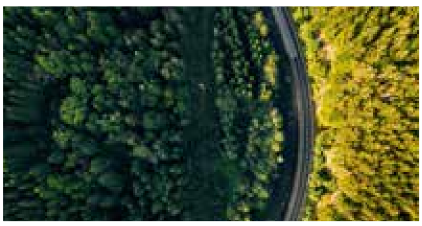
Year 17 n°3

Autumn 2020

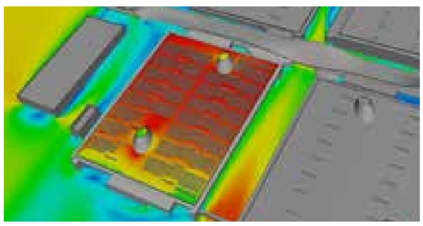
Using CFD Analysis to predict the effect of Wind on a Tower Crane



The spread of pandemics and the role of numerical simulation - interview with Thierry Marchal



Real-time route optimization of a drone: people detection and mission reconfiguration

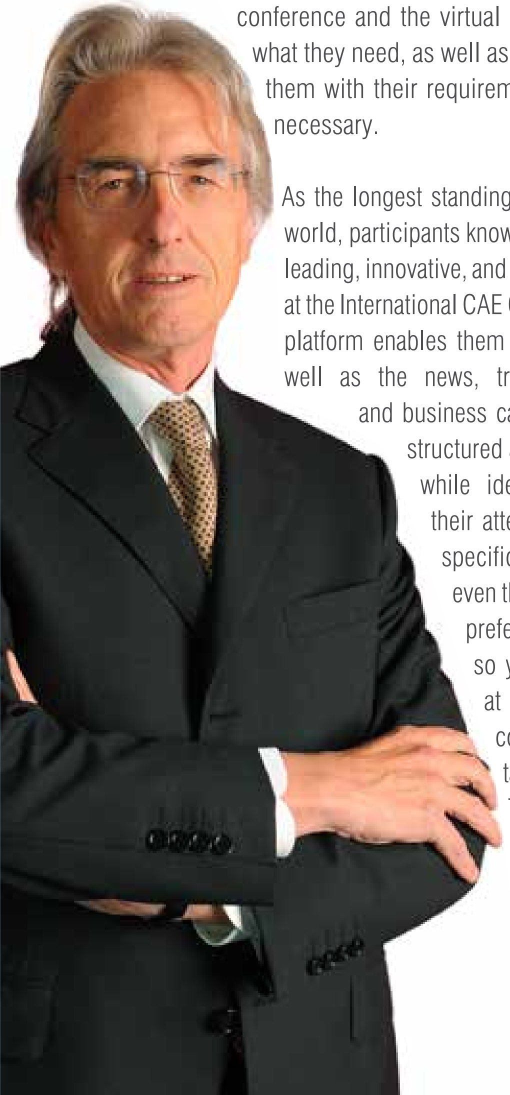


Placing computational wind engineering at the fingertips of structural engineers

Flash

The decision to convert the 2020 International CAE Conference and Exhibition – the 36th time this annual event is being held – into a virtual event, from November 30 to December 4, 2020, was driven by the dramatic and continuous evolution of the COVID-19 [https://www.cdc.gov/coronavirus/2019-ncov/faq.html, accessed 29/09/2020] disease across the planet. The new virtual platform offers exciting digital advantages to participants – exhibitors, speakers, and attendees – not least of which is the real ability to reach an authentic international audience, making it possible for all stakeholders to benefit.

With exhibition and conference presentations being broadcast in three time slots, event content will reach audiences in the Asia Pacific, Europe and Africa, and the Americas at convenient times to facilitate their participation. For the proponents of technologies or services, this means the ability to reach their own reference market, but also new markets of potential partners and customers that may have been out of reach previously due to geographical or economic barriers. Participants and attendees meanwhile will have access an extremely wide range of presentations from both the conference and the virtual exhibition in order to find what they need, as well as who may be able to assist them with their requirements – closer to home, if necessary.



As the longest standing CAE-specific event in the world, participants know that they will find thought-leading, innovative, and scientifically sound content at the International CAE Conference. The new digital platform enables them to access this content, as well as the news, trends, technology-specific and business case information in a highly structured and content-rich format, all while identifying and customizing their attendance according to their specific interests and needs, and even their geographically specific preferences. If you haven't done so yet, visit the event website at www.caeconference.com to register and begin tailoring your participation. There are still ample opportunities for vendors and service providers, too.

An important reason people attend conferences and exhibitions is to keep abreast of new developments, which often reveals gaps in our knowledge, or new technologies or techniques to be learned and mastered. The sourcing of specialized or multi-skilled talent, the ongoing challenge of continuous skills development, as well as the acquisition and refinement of the soft skills so critical in business today, are universal challenges faced not only by companies, but also by individuals.

To begin to address some of these requirements, EnginSoft is busy overhauling its entire training offer. Again, in response to the pandemic, all training has been moved online, although it is still possible to participate in training courses in person, pandemic-related circumstances permitting. Moving online has meant that a significant portion of these training courses that are in English can be accessed and used worldwide, everywhere from India to Chile, so that individual engineers or companies in the engineering fields can take advantage of EnginSoft's world-leading technology and simulation application experience wherever they are.

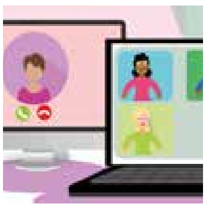
The company is also able to organize customized training for engineering organizations, either in a vertical sector, or on a specific software technology in use. Again, this training is available either online, or onsite at the company when circumstances, regulations, and restrictions permit. EnginSoft's training solutions are being reorganized to move away from a focus on software programs and more towards a focus on the techno-scientific engineering disciplines, such as mechanical, fluid dynamics, process integration and multidisciplinary optimization, and others.

While software training will still be available within each of these application areas, the new focus on scientific application areas will enable the company to offer a deeper understanding of the technological and industrial trends and developments in each area, as well as enable new graduates, or engineers specialized in other areas to study in different areas, with the intention of helping to break down the silos of knowledge in organizations and facilitate greater understanding of the different processes and challenges up- and downstream of an engineer's specific role. Keep an eye on this space for further information in the following months! In the meantime, information about the courses currently available can be found at: www.enginsoft.com/training/

Stefano Odorizzi,
Editor in chief

Contents

Case Studies



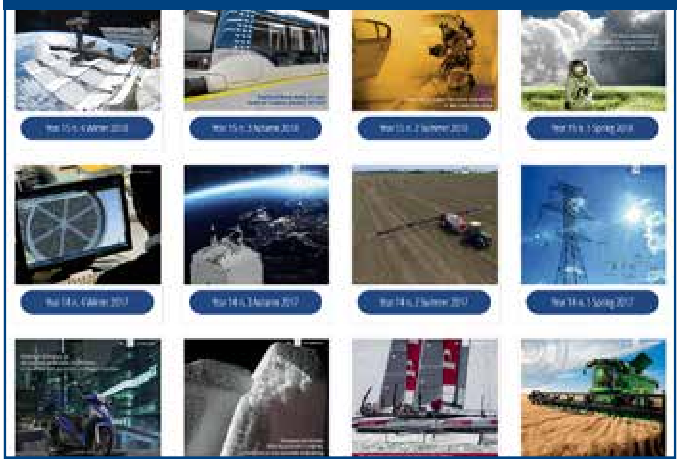
Interview

The spread of COVID has hit the population of the entire planet hard, demonstrating concretely and dramatically the weaknesses of a globalized and interconnected society. Science can do a lot to counteract and prevent events of this magnitude. To give readers of Analysis and Calculation a perspective on the role of simulation and computation we interviewed Thierry Marchal, Global Industry Director for Healthcare at Ansys, a multinational company operating in multi-physics digital simulation.

Research & Technology Transfer

Software Update

PAST ISSUES at
www.enginsoft.com/magazine



Newsletter EnginSoft
Year 17 n°3 - Autumn 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

By Marco Evangelos Biancolini
Università di Roma "Tor Vergata"

The spread of pandemics and the role of numerical simulation - interview with Thierry Marchal

The spread of COVID has hit the population of the entire planet hard, demonstrating concretely and dramatically the weaknesses of a globalized and interconnected society. Science can do a lot to counteract and prevent events of this magnitude. To give readers a perspective on the role of simulation and computation we interviewed Thierry Marchal, Global Industry Director for Healthcare at Ansys, a multinational company operating in multi-physics digital simulation.

Marco Evangelos Biancolini - I would like to exchange some ideas about what is happening in the world and the role that numerical simulation can play in fighting pandemics. What is your point of view on the subject?

Thierry Marchal - I believe that today more than ever simulation can play a key role. There are two main categories to focus on: the first is prevention and the second is treatment. Prevention is related to social distance, to PPE (personal protective equipment), in particular the mask. In this case, simulation can help to better understand what happens when jogging or walking fast. Walking fast could be dangerous for someone who is walking behind or close to you because you cannot control the particles that escape during breathing. There is some disagreement about social distancing: governments suggested one meter, two meters, in some cases even ten meters, and this confuses many people. But what is the reality? Often, even experts provide contradictory information, increasing confusion. The beauty of simulation in this respect is that it cannot only support research, but also contribute to social education. As soon as people see with their own eyes how particles propagate in different everyday situations and understand that the spacing indications depend on the specific situation, they will act accordingly without any impositions.

Related to this, a few months ago I remember that I first received the results of a simulation carried out by Professor Yu Feng of Oklahoma State University. The simulation showed the propagation of saliva droplets after a cough between two people at one meter and 1.8 meters apart. My 11-year-old daughter was watching the video and said, "OK, Dad. I cannot stand 1 meter away; I feel like it is too close. I'll keep 2 meters away from anyone."

Another appreciable result, besides education, is training: the results obtained from simulations are useful for training medical staff. For example, if a patient is hospitalized suffering from COVID, regularly a patient mask change should be operated by the staff. However, do surgical masks worn by the nurses protect enough against the risk of contaminated particles entering laterally and flowing under the nurse mask with a risk of contamination? Modeling the contact between the mask and the medical staff face is then very useful to evaluate both containment capacity and comfort: after many hours, a patient or staff's face may become inflamed due to the excessive pressure of the mask.

Simulation can also address another important aspect that is the contamination of surfaces when travelling by plane or train. There are several possible solutions to this problem and once again simulation can help by providing indications, for example, on delivering the right doses of UV per cm2 (fig.2) to decontaminate every surface touched by passengers. In this instance a robot that uses UV rays can be used and by simulating the areas actually irradiated it is possible to ensure that the surfaces are decontaminated quickly, efficiently, and effectively. For example, the decontamination of a plane or a train could take place within the time passengers are waiting to board. Since this does not require that many people to decontaminate the vehicle, a good compromise can be achieved between cost for the company and safety for the passengers.

At the moment, however, the big struggle to beat COVID is mainly about pharmaceuticals. Researchers are working to find a vaccine and it is hoped that this will happen between late autumn and winter in Europe. As soon as a vaccine is approved it will have to move to large-scale production with a manufacturing process capable of producing vaccines for literally billions of people: never before in human history have there been any production on this scale. Simulation can once again play a key role to address this challenge.

One could also talk about classrooms, restaurants, gyms, or operating theatres and explore the possibilities of changing the geometry of air exchange systems. For such complex environments safe solutions can be supported by "digital twins" based on numerical fluid dynamics and mesh morphing tools. This approach is already being used for healthcare facilities. In neonatal intensive care units, the preparation time for personnel to properly dress before accessing newborn babies safely and sterilely can be up to 45 minutes. This is far too long for an emergency and could be fatal in some cases. How can this time be reduced? Simulation can be used to solve the problem. The ventilation system can be modified to enable emergency access by personnel which have not performed additional sterilization while not compromising with the safety of the patient. Contamination in this case is prevented by adjusting the air exchange system so that patients are protected by invisible air barriers.

M.E.B. - A rapid solution based on fluid separation. So, the basic idea is precisely controlling the HVAC?

T.M. – Certainly, but the filters require special attention because the air that is sucked in may contain viruses. Also, coming back to the case of restaurants, gyms, and other everyday environments, the solutions must be fast, cheap, and easily accessible. One approach could be to create easily adaptable templates of lifestyle spaces. You can empower a restaurateur with an online tool to assess his or her own table layout and the optimal adjustment of the air treatment system. So, the keywords are: inexpensive and quick solutions!

M.E.B. - What do you think of the BIM protocol for the digitalization of buildings? Do you think we could insert an active control for an HVAC to have a better air quality?

T.M. – This has been the dream of BIM since the beginning. In the future we will have digital twins not only for health safety but also for energy. It will be possible to regulate the temperature and humidity level in houses and save a lot of energy. I think that despite its negative aspects related to the death of so many people, COVID could be an opportunity to smartly invest in health safety and to improve people's behavior. Simulation using a digital twin, combined with the BIM approach for buildings, could also help us in the future, drastically reducing the spread of influenza or any other type of virus. So, I think this could be an advantage to be used but only if we recognize the central role of simulation.

M.E.B. - We talked about restaurants, planes, running outdoors. But how should people behave when practicing sport indoors?

T.M. –We have just published an article on gyms that take advantage of ventilation from above to push air down using air vents in the lower part of the building. A similar numerical approach was already used years ago when new buildings were constructed. The important factor was to ensure that the new building would not adversely affect the comfort of outside pedestrians due to the different way in the wind was channeled.

The use of simulation also makes it possible to optimize interior spaces. At the Ansys data center in Germany, we were able to reduce energy consumption by working on the layout and positioning of the server cooling systems, thereby saving EUR 10,000 in annual costs. (https://www.ansys.com/-/media/ansys/corporate/resourcelibrary/article/aa-v5-i2-better-cooling-hot-savings.pdf).

M.E.B. - I understand that you envision a possible role for simulation once we have a vaccine and have to produce it. What is it?

T.M. – Whenever you work on new drugs or vaccines, you do not use very large mixing tanks or large bioreactors. The dimension is small because you produce small volumes of vaccine in the laboratory to be tested with a few people only. Even for a few thousand patients you are still on a pilot scale with mixing tanks of a few tens of liters. For COVID we will need a very large-scale production and to go from one liter to 5000 liters and beyond, simply scaling all sizes by a factor of 10 or 100 is not sufficient: we need to make sure the blend is the same quality no matter the size of the equipment. Here the experience

Fig. 1 - Social distancing

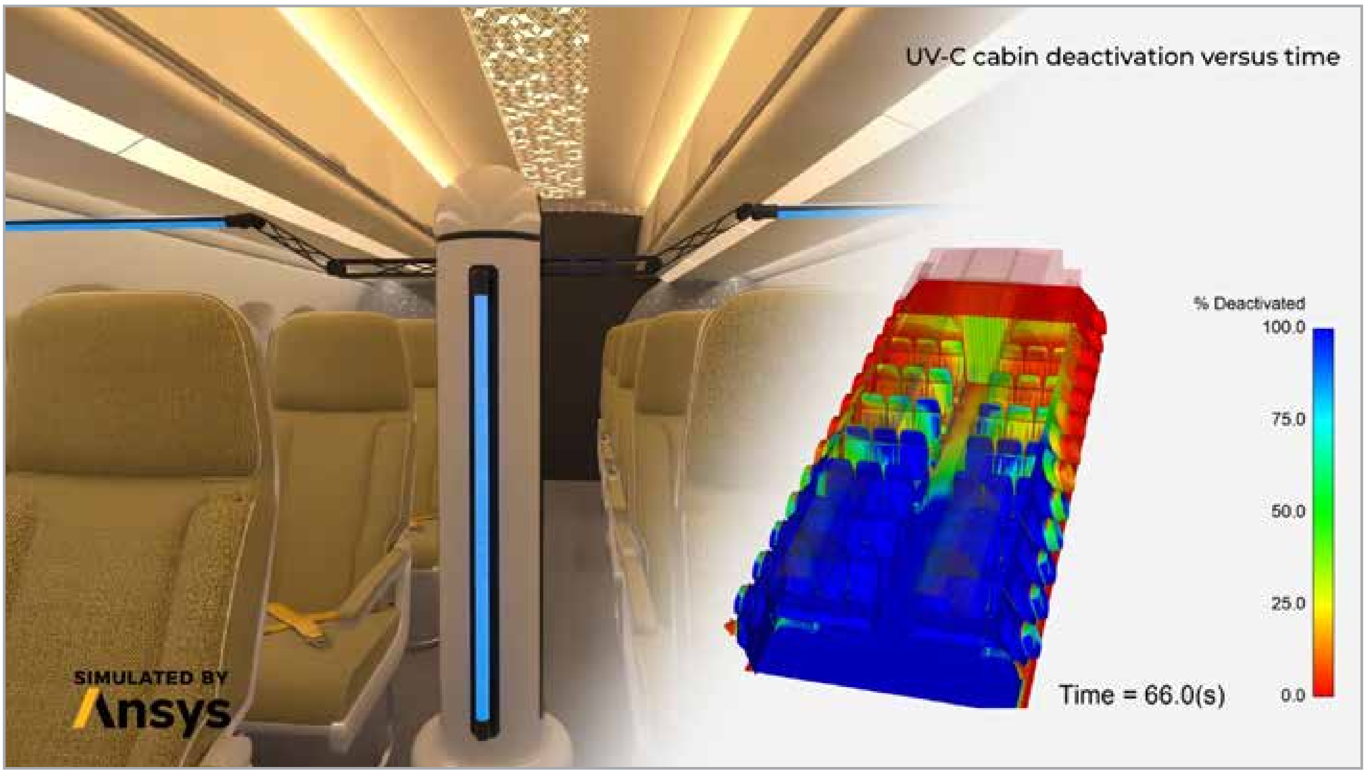


Fig. 2 - Snapshot of irradiation (W/m2) inside a cabin using a UVC lamp system.

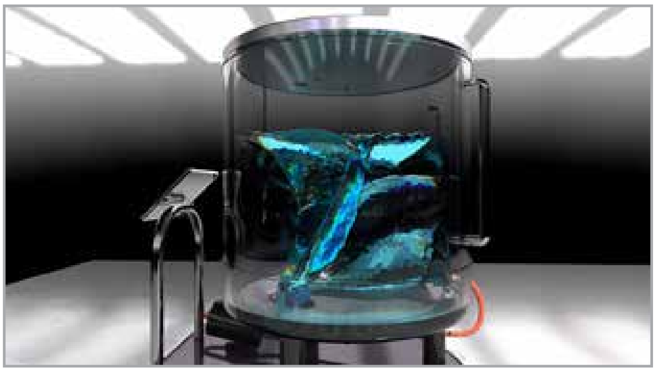


Fig. 3 - Simulation of the mixing process in a bioreactor.

of decades of simulation plays a key role in better understanding the mixing process and in creating bioreactors that can guarantee quality even for mass production (fig.3).

To respond to the emergency, our company is committed to making available all necessary and appropriate resources in terms of both simulation technology and skills.

M.E.B. - I have one last question. When we consider the option of customizing an HVAC with cloud-based tools, using the standard tools of Ansys or others, which TRL should be chosen? Can we trust these tools to counter the propagation of the virus? Are they ready to be certified?

T.M. - The primary point that we are addressing with regulatory authorities from around the world is that every solver and model have to be verified and validated. A long collaboration involving leading healthcare companies including Ansys led to a description of the steps to follow to verify and validate a solution for FDA approval; this is detailed in a document published by ASME (the V&V 40 document). Similarly, we are now working closely with the European Medicines Agency (EMA), on a " white paper" on similar topics but with a focus on pharmaceuticals; unfortunately regulations in the various countries are still different, we need to harmonize them. All the certifying bodies are aware of the importance of simulation, what we also call in silico methods by comparison with in vivo and in vitro approaches, but at the same time they want to be sure to define the right guidelines to ensure that any type of model is properly validated.

We are working with governments around the world, including the Australian authorities, to regulate the criteria governing the acceptance of simulations. There is a lot of work to be done. We are working with agencies as well as governments who have to provide guidance and adapt legislation so that simulation-based approaches can be used with the goal to drastically accelerate medical innovation. This is one of the key roles of the Avicenna Alliance for whom I have the privilege to serve as Secretary General.

M.E.B. - Thanks Thierry. It has been an extremely pleasant conversation; I am sure readers will find it interesting.

T.M. - It has been a pleasure Marco. I would like to inform readers that Ansys has set up a page on its website dedicated to the subject. It is regularly updated, and you can find out more about a lot of the news I mentioned today (ansys.com/COVID).

Pictures by courtesy of Ansys
A special thanks to "Analisi&Calcolo" magazine which first published the interview with Thierry Marchal in its 98th edition. (May/June 2020).

Using CFD Analysis to predict the effect of Wind on a Tower Crane

CFD proves to offer a reliable working methodology for designers

By Francesco Valente[1] and Alessandro Arcidiacono[2]
1. Terex Cranes - 2. EnginSoft

Designers of tower cranes must consider many variables when translating the interaction between air flow and a structure into an integral force. Synthesizing the variables makes the calculations more precise and easier to perform, without compromising safety margins. Industry regulations must also be followed in creating appropriate predictive models for evaluating the effect of airflow on these cranes. The reference standard for calculating the wind loads acting on a tower crane are conceptually simple and easy to implement and include many rules and parameters that can help designers to evaluate the shielding effects of components' shapes. However, engineers must typically be guided by their intuition and experience to understand how much one element is shielded by another and how intensely they are affected by the air flow. This article proposes the use of computational fluid dynamics (CFD) to address these complexities.

The design of a tower crane is composed of two consecutive, mutually dependent phases. The first phase involves the development of the machine itself; the second concerns its application in construction sites. When sizing the parts of the tower crane, the designer must take into account a series of loads. One of the most important of these, due to its insidiousness, is the effect of the wind on the exposed surfaces, which is a complex phenomenon that must be addressed during the design phase.

There are many variables to consider when translating the interaction between an air flow and a structure into an integral force. Therefore, it is essential to synthesize the variables in order to make the calculations more precise and easier to perform, without compromising safety margins. Moreover, it is vital for the design engineer to follow industry regulations in creating an appropriate predictive model for the crane.

The traditional approach

The reference standard for calculating the wind load acting on a tower crane is FEM1.001, which has particularly easy settings due to its conceptual and implementation simplicity. This calculation method assumes that the wind is acting horizontally, that it can blow against the tower crane from any direction at a constant speed, and that it is generating a static force.

Dynamic pressure is thus calculated with the well-known formula:

- $q = (0.613)v^2$ (1)
  where "v" is the velocity of the air flow.

To determine the force caused by this pressure on a structural element, the following expression is used:

- $F = q\,A\,c$ (2)
  where "q" is the dynamic pressure derived from (1), "A" is the protrusion of the surface area hit by the air stream, perpendicular to it, and "c" is the shape factor of the covered surface.

The standard includes a table of the form factors for the most common profiles used in carpentry construction. In order to determine the resulting force on the whole machine, the force acting on the individual components is calculated using (2) and, finally, the total sum is calculated.

The last step of the process, which is to evaluate the shielding effects that a generic element will have on the next one, is perhaps the most critical and can place the design engineer in difficulty. While the standard has many rules and parameters that can help the designer to evaluate these shielding effects, engineers must typically be guided by their intuition and experience to understand how much one element is shielded by another and how intensely they are affected by the air flow.

Computational fluid dynamics vs the traditional approach

As mentioned above, the method proposed by the FEM1.001 standard allows engineers to easily perform a numerical evaluation of the wind force on a crane, using a completely traditional calculation.
However, some critical aspects persist, such as:

Fig. 1 - Terex flat top CTT172-8 crane

- The accurate prediction of the shape coefficient of a given profile
- The evaluation of the shielding effects of one element on the next

These issues represent a real challenge for the engineer performing the calculations. The computational fluid dynamics (CFD) calculation discussed below, therefore, presents an innovative attempt to overcome these uncertainties and to assess the feasibility and accuracy of the traditional approach.
The Ansys CFD analyses were performed for a Terex flat-top CTT172-8 crane (Fig. 1), according to the FEM1.001 standard mentioned in paragraph 2, and the resulting force was used as a reference for the comparison between the two methods.

The CFD model

Geometry and mesh

A CAD model of the tower crane was supplied by Terex: the geometry consisted only of the upper part of the tower crane, specifically the operator cabin, the concrete counterweight and the jib. Three different crane lengths were considered: 65, 60 and 55 meters (Fig. 2).

The crane lengths were obtained by “cutting” the farthest part of the jib, while the counterweights were reduced accordingly.

To accurately calculate the fluid dynamics surrounding the crane, a suitably large external domain was considered in the CFD model. The dimensions of the box in this case were 1.0 x 0.8 x 0.2 km (Fig. 3).

Different wind directions were evaluated by simply rotating the cylindrical domain of the crane relative to the wind inlet. Clearly, the major challenge of this CFD analysis concerned the over dimensions of the geometry, together with the extremely detailed CAD of the solid parts of the crane. Small bolts, screws, and panels were common and the geometric anomalies within the CAD were too long and cumbersome for the user.

Fluent Meshing’s fault-tolerant mesh workflow was crucial to successfully mesh the fluid geometry, preserving the relevant part shapes of the crane while neglecting all other details. Thanks to this powerful tool a triangular surface mesh of the tower crane was created, from which it was possible to extract the fluid domain directly.

A tetrahedral and prism approach was then used to mesh the cylindrical section of the tower crane, while the external box was meshed using a structured strategy. The resulting meshes have very large numbers of elements, due to the overall dimensions of the model and the level of mesh detail (Fig. 4).

Solution and results

The Ansys CFD model was essentially configured for an external aerodynamics problem with incompressible, turbulent, steady state conditions.

An air temperature of 20°C was selected for the working fluid, with a constant wind inlet velocity of 20 m/s. Three different wind directions

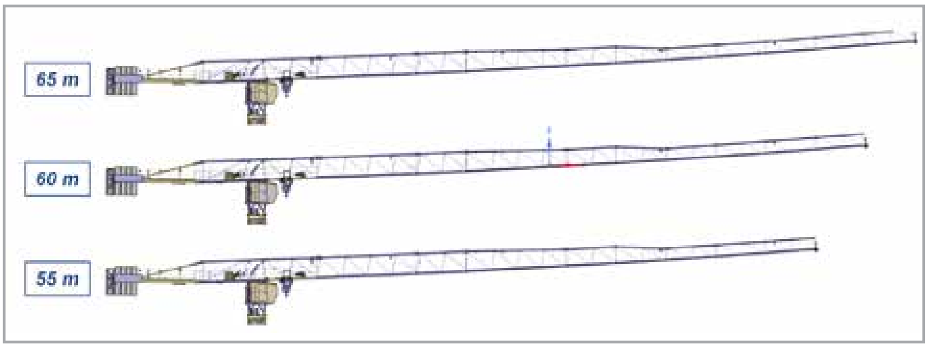


Fig. 2 - CAD models

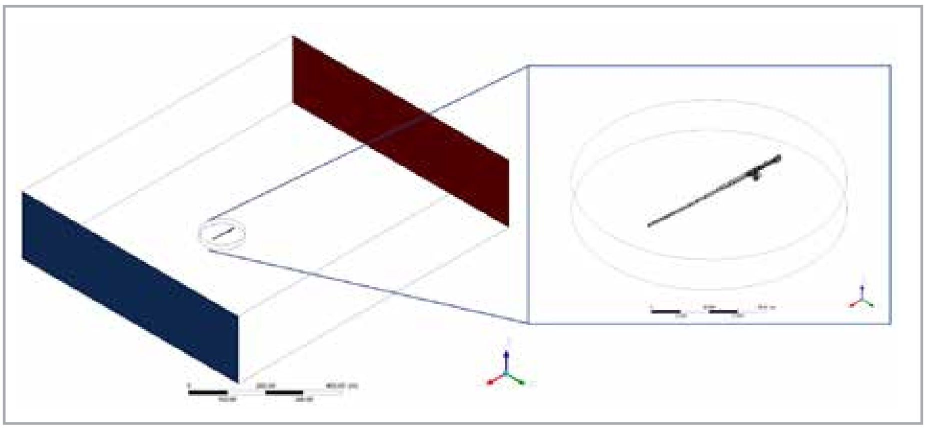
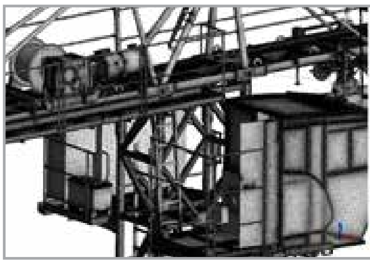


Fig. 3 - CFD external domain, with Inlet (blue) and Outlet (red)



Tower Crane length	# Mesh Elements
65 m	103,694,588
60 m	104,029,346
55 m	97,927,505

Fig. 4 - Mesh

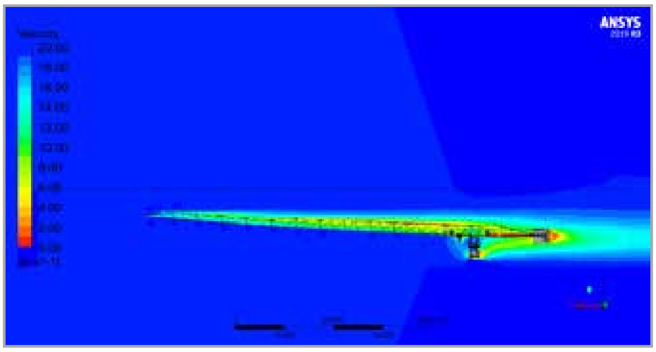


Fig. 5 - Velocity contour on the longitudinal section

were considered: headwind (front), side and tailwind (rear), which were achieved by rotating the cylindrical domain of the crane. The k-omega shear stress transport (SST) turbulence model without ground contour effects was used. Free flow (symmetry) conditions were used on the sides of the bunding box.

Results were both qualitative and quantitative, in terms of the pressure and velocity contours (Fig. 5) and overall pressure on the crane surfaces. The shielding effect of the wider surfaces on those that follow is evident in Fig. 5.

Terex was mainly interested in predicting the resulting forces along the three Cartesian directions: longitudinal (X), vertical (Y) and transverse (Z). Fig. 6 shows the cumulative plots of the X (longitudinal) force for the case with headwind. The similarity of the results for the three cases is noticeable, with a slight increase in overall force intensity from 55 to 65 meters, in line with expectations. It can be also noted that the operator's cabin and the slewing unit are responsible for most of the resulting overall force, while the jib has a much lower effect on the force per unit length.

Conclusions

The CFD study focused on the forces acting in X, Y and Z directions on the Terex tower crane, with three jib length configurations, and one wind speed from three directions.

The results were in very good agreement with each other, especially for the headwind and tailwind cases, where the overall X force was almost the same in intensity but opposite in direction.

The differences in the tower crane lengths seemed not to be so relevant, since the overall X force could easily be estimated from one result for the other tower crane lengths. The results of the CFD calculation were very close to those obtained using the traditional methodology: the integral force forecast was in line with Terex's expectations. More

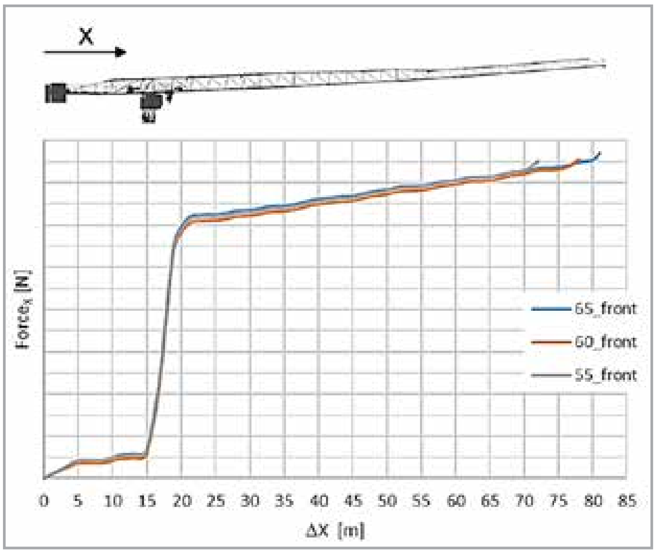


Fig. 6 - Longitudinal force cumulative plots, Front Wind Case

detailed information is provided by the CFD local force forecast, however, especially where surfaces with greater extension offer significant shielding effects.

In conclusion the CFD analyses yielded overall results that were comparable to strict compliance with the standard's provisions for shape factor selection, determination of surfaces exposed to wind, and shielding effect, but provided a more detailed insight into local phenomena, both qualitative and quantitative, thus furnishing the designer with a reliable working method.

About Terex

Terex Corporation is a global manufacturer of lifting and materials processing products and services that deliver lifecycle solutions to maximize customer return on investment. Major Terex brands include Terex, Genie and Powerscreen.
Terex solutions serve a broad range of industries, including the construction, infrastructure, manufacturing, shipping, transportation, refining, energy, utility, quarrying and mining sectors. Terex Cranes, part of Terex Corporation's Material Processing segment, includes the Rough Terrain Cranes and Tower Cranes businesses. Their broad product range includes rough terrain cranes, self-erecting, flat top, luffing jib, and hammerhead tower cranes.

References

[1] FEM1.001: "Rules for the design of hoisting appliances". Wind forces are dealt with in Booklet 2: "Classification and loading on structures and mechanisms".
Since this article is not intended to describe the regulatory framework governing the design of a crane, the reference will be limited to this regulation only. In the usual design process, other standards, in addition to FEM1.001, are also used to assess the wind effects.

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

Filling and compression analysis of metallic powders composed of spherical particles

The efficiency of material design can be greatly improved by using FEM-based virtual material testing analysis

By Koji Yamamoto
CYBERNET SYSTEMS CO.

This article presents an example of the analysis of a powder composed of spherical particles. Since powders have a very fine heterogeneous structure, a multi-scale approach based on homogenization analysis is proposed. The macroscopic material behavior of the powder is obtained through virtual compression tests using a unit cell model representative of the microstructure. The microstructure is further homogenized by curve-fitting its material response according to the constitutive law of the macro material. The analysis was performed using Ansys® MechanicalTM and its add-on tool Multiscale.Sim.

Metallic powders are used to create new materials for many industrial manufacturing processes. Any number of metals can be used in the mixture to change the characteristics of the final product, in terms of strength, flexibility, durability, etc.

For a material with a large number of particles, particle densification is very important because there is a correlation between density and the material properties. For example, the electrodes used in rechargeable batteries consist of densely packed particles (also called as active materials). The electrons inside the individual particles are charged. This means that the electrical properties, such as battery capacity and charging efficiency, are closely related to the particle density. However, the mechanical properties also have an influence, so it is equally important to understand the mechanical properties of the product in order to improve its reliability and durability, and so on.

There are many conditions involved in creating a material full of particles, such as the shape and size distribution of the particles, and the type of material. Therefore, the material properties are not easily optimized using only actual testing; rather, it should be combined with simulation.

Many researchers have analyzed the process of particle filling. Most of them used the discrete element method (DEM) [1], [2]. The DEM was considered highly compatible with these analyses because it can efficiently calculate the motion of individual particles, which are assumed to be rigid. The inelastic behavior that occurs during the compression process is then represented by the interaction between the particles. However, since these parameters are not directly related to real phenomena, it is necessary to calibrate them using actual tests. The finite element method (FEM), on the other hand, can directly express the deformation behavior of particles, which allows us to analyze phenomena that are closer to the real problem.

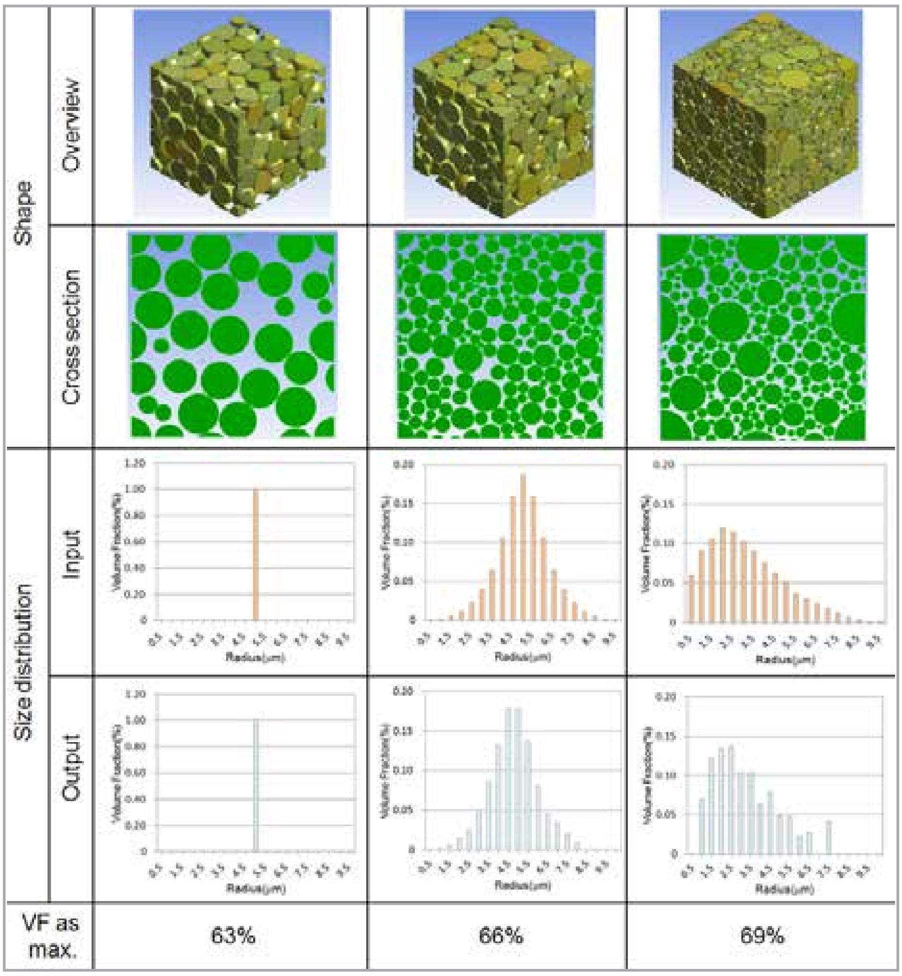


Fig. 1 – Microstructure created for three types of size distribution before compression. Maximum volume ratio increases as the number of small particles increases.

This article presents an example of analyzing the particle packing problem using the finite element method. First, I will discuss the issue of particle packing to establish the initial state before compression. Multiscale.Sim™ [3], an add-on tool for Ansys® Workbench™ for multiscale analysis, was used to create the model. Next, an example of the analysis is shown whereby a rigid wall is placed around the micromodel created and then densified by compression.

Here, we will observe the change in the microstructure during compression and the relationship between the compression force and the fill rate. Finally, the resulting macroscopic material behavior is adapted using the macrostructural constitutive law. With the equivalent material properties obtained in this way, you can perform the analysis of the macrostructure using the homogenized microstructure.

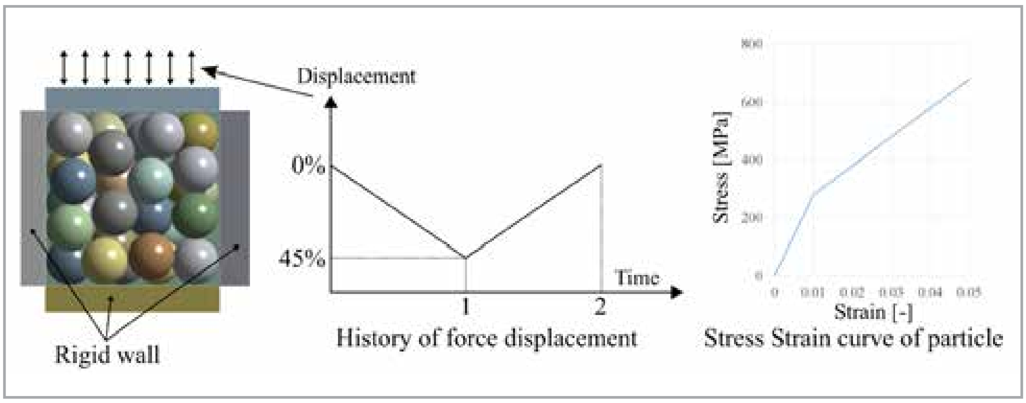


Fig. 2 – Overview and conditions for the virtual material test of the compression analysis.

1. Particle filling analysis

Before presenting an example of the analysis of the compression process, we will consider the initial composition of the particles in the powder. The initial placement of the particles is very important because, if there are multiple bullet gaps or bites between particles in the microstructure from the beginning, it can compromise the accuracy of the compression process analysis. The particles should be packed with as few gaps and bites as possible. By observing the initial packing state of various distributions of particle sizes before compression analysis, it is also possible to predict trends in the maximum particle packing rate after compression.

As a result, we adjusted the distribution of the particle sizes considered here for three different types (uniform, gaussian and bias) and used Multiscale.Sim to fill the particles to the limits of the instrument for each condition. Each condition was defined so that no overlap was allowed between the particles. Fig. 1 summarizes the appearance and particle size distribution of the model. If the number of particles that make up the microstructure model is too small, it is not possible to create a model with a specified particle size distribution.

Therefore, we adjusted the size of the unit cell so that about 3,000 particles could be created. Consequently, we confirmed that it is possible to create a model similar to the input size distribution.

A solid model with a uniform particle size was found to have a volume content of about 63%. Generally, we know the theoretical maximum volume content is 74% when the particles are regularly aligned. On the other hand, however, when the particles are randomly scattered, the upper limit of the theoretical volume content is considered to be 64%, which is almost identical to the results obtained by Multiscale.Sim.

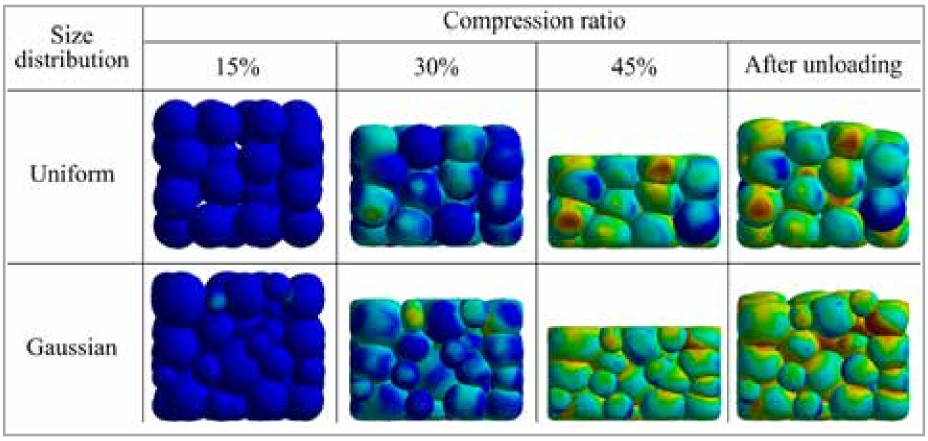


Fig. 3 – Deformation shape and equivalent plastic strain during compression

processes were configured to take 1 second each. These analysis conditions are shown in Fig. 2.

Fig. 3 shows the deformation shapes of the microstructure and the distribution of the equivalent plastic strain at various points in the compression process. The analysis was performed on cases with no size distribution and cases with a Gaussian distribution. The displacement constraints defined on the rigid walls created conditions that were close to hydrostatic pressure from a macroscopic point of view. Large plastic strains can be observed inside the microstructure despite the fact that the particles do not yield due to hydrostatic stress. This means that the heterogeneity of the microstructure generates a complex stress field within the microstructure, even though the macroscopic conditions are hydrostatic.

The results demonstrate that Multiscale.Sim is capable of packing the particles to the limit. In the case of size distribution, the spaces created between relatively large particles are filled with smaller particles, so a larger volume content can be achieved than in the case of particles of uniform size. It should be possible to create an even higher volume content by increasing the probability of the presence of statistically small particles. The results of this analysis are consistent with those considerations.

2. Compression analysis

The compression process analysis for densification is performed using the particle-filled microstructure model created in the previous section. The microstructures analyzed in this study are the result of extracting only a small portion of the actual structure. In order to evaluate the apparent behavior of the material based on such a partial model, the homogenization method is generally used.

In this analysis, the extracted microstructure is used as a unit cell, and conditions of periodic symmetry are defined in all directions on its outer surface. However, if the particle-free region is not filled with other materials, such as resin, it is inevitable that small particles will escape from the unit cell during compression.

Therefore, an alternative is to create a rigid wall that touches each of the six outer surfaces of the unit cell. One of these rigid walls is then loaded by forced displacement and the unit cell is compressed. The compression displacement was applied to about 45% of the macro strain and then returned to its pre-compression position. The other five rigid walls were fully constrained by the displacement degrees of freedom. The elasto-plastic properties of the Mises yield function were defined for the individual particles. The friction coefficient for all interfaces between each part was set to 0.3.

These models were used to perform an implicit analysis using Ansys Workbench. In order to ensure the stability of the analysis, a transient analysis was conducted to take into account the effect of inertia. For the loading history, the compression and unloading

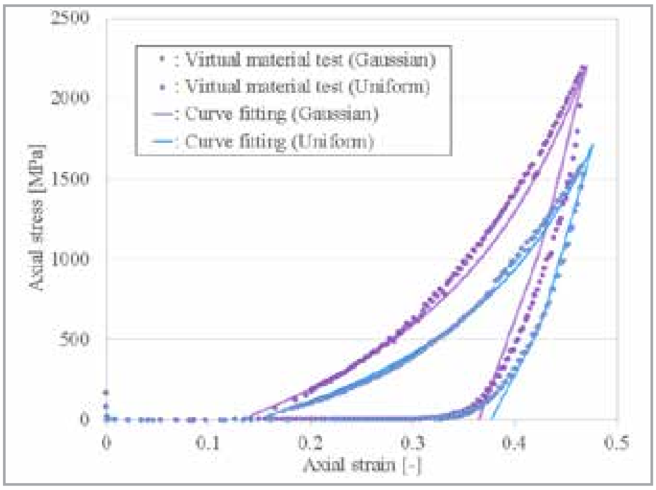


Fig. 4 – Macroscopic stress strain curve as a result of the compression analysis

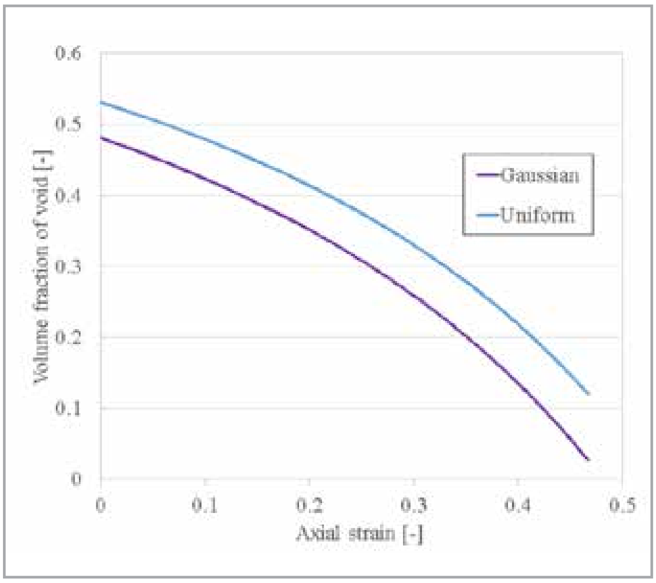


Fig. 5 – History of the volume fraction of the void in the microstructure during compression

Fig. 4 shows the macroscopic stress-strain characteristics calculated from the reaction force of the rigid wall under compressive displacement and the history of porosity in the unit cell. In the compression process of such porous materials, the tangential gradient tends to increase rapidly with increasing strain. This is due to the fact that the voids inside the microstructure are reduced by compression as shown Fig. 5.

This reduces the inhomogeneity and brings the inside of the microstructure closer to the same hydrostatic pressure conditions as the macroscopic external force.

In this paper, the accuracy could not be verified because the actual measurements were not made under the same conditions as the analysis. However, since the stress-strain shape shows the same tendency as the generally acknowledged facts, it is qualitatively considered to have captured the actual behavior to some extent. In addition, as shown in fig. 4 and fig. 5, we believe that the changes in the inelastic material behavior during press forming due to differences in particle size distribution qualitatively capture the correct tendency.

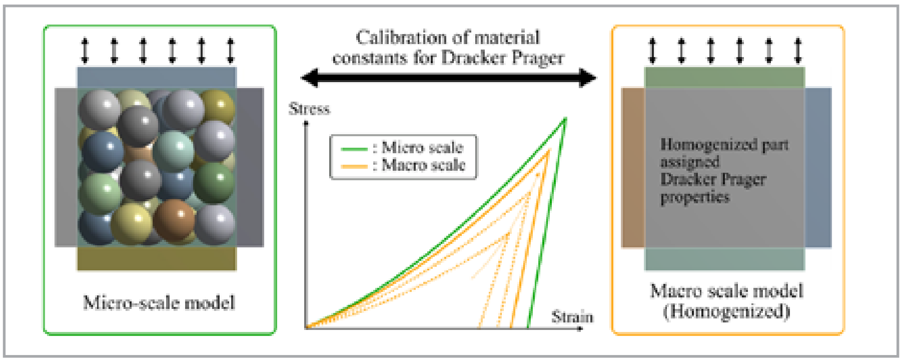


Fig. 6 – Analysis flow to identify the material constants for macro scale analysis

3. Curve fit analysis

Up to this point, we have analyzed the material behavior during the compression process using a microstructure model composed of particles.

However, this approach is computationally expensive and can only focus on extremely limited areas for analysis.

Modeling for heterogeneity is not appropriate for actual product-level objects. Therefore, at the end of this article, we attempt to replace the heterogeneous microstructure with a homogeneous body.

To do so, the behavior of the macroscopic material obtained in the previous section must be adapted to the material composition radius shown in Fig. 4. Powdered materials composed of particles show remarkable elastoplastic properties even under hydrostatic conditions. These properties cannot be expressed by the Mises yield function, where deviation stress triggers the generation of plasticity.

In this paper, Drucker Prager's capped model [4] was used. This is a typical constitutive law for materials to represent the yielding behavior dependent on hydrostatic stress (accessible with the command "TB,EDP,,,,CYFUN" in Ansys software). Fig. 6 shows the curve fit analysis flow. The homogeneous blocks defined according to the Drucker Prager constitutive laws were compressed by the same boundary conditions as in the previous section. The material constants were adjusted so that the stress-strain properties obtained in this analysis correspond to the response of the heterogeneous microstructure model analysis evaluated in the previous section. Ideally, this adjustment work should be done by optimization analysis, but in this case, it was performed manually. In future, we plan to implement a fitting algorithm based on the optimization analysis in Multiscale.Sim products.

The result of the fitting is shown as a solid line in Fig. 4. There is a slight error in the results. However, the tendency for the stress to increase exponentially with increasing plastic deformation, and the tendency for the unloading curve to move downwards in a convex manner, are well captured. The effect of adjusting the material properties to change the particle size distribution is also well represented. Although the fitting results may not yet be sufficient, we do not think that this is a problem caused by the ability to express the constitutive law of the material. We believe that more accurate fitting will be possible in future by incorporating an optimization algorithm.

Conclusion

The results of the analysis showed that the material behavior during compression, and the degree of densification that can be achieved by compression, can vary greatly depending on the size distribution of the particles. Because of the wide range of parameters that determine the material behavior of powders, it is not realistic to design materials based on actual measurements alone. It is expected that the efficiency of material design will be greatly improved by using the virtual material testing analysis introduced here.

**Koji Yamamoto is Technology Specialist and Developer of Multiscale.Sim at Cybernet System Co.,Ltd.*

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

References

[1] Subhash C. Thakur, Hossein Ahmadian, Jin Sun, Jin Y. Ooi, An experimental and numerical study of packing, compression, and caking behavior of detergent powders, Particuology, Vol.12 (2014).

[2] Yi He, Tim J. Evans, Aibing Yu, Runyu Yang, Discrete Modeling of Compaction of Non-spherical Particles, Powders & Grains, Vol.140 (2017).

[3] https://www.enginsoft.com/solutions/multiscale.html

[4] T. J. Watson, J. A. Wert, On the development of constitutive relations for metallic powders, Metallurgical transactions, Vol. 24A (1993).

Optimization techniques applied to centrifugal compressor design

Using RSM to save design time and speed time-to-market

By Ileana Vitale
Ingersoll Rand

During the development of a centrifugal compressor stage, different challenges arise if the focus is not only on the best efficiency, so critical nowadays. Time-to-market is also critical and once a new product is created, sales support and the improvements feasible during the product's life are also subject to time constraints. Computational time has been reduced significantly compared to the past due to latest algorithms and information technology. Carefully analyzing the available tools it is possible to combine them to optimize the process flow and use them in an "unconventional" way. Optimization tools, and specifically response surface methods (RSM), can be adapted very well in the design process to provide information around the design of a compressor stage. This article will cover two of the possible optimization uses: the search of optimum performance and data generation.

The design of a centrifugal compressor, as any other industrial equipment, is a specific process that involves a variety of disciplines. The most relevant aspect is the level of interconnection between the various technical features, to the point that iterative design processes are on the agenda of every engineer and designer. These processes can be implemented within the single component of the design's characteristic or within the same project.
The incredible scenario of available software helps creating easy connections and data exchange and is also automating most of the iterative processes.

Optimization is a now growing process supporting design activities. An interesting perspective offered by optimization is the possibility, within the same analysis, to connect various design aspects into a single "box" to achieve a unique objective. This objective is actually shared by all the technical features involved and that are represented by the different engineering disciplines.
Several different centrifugal compressor challenges can be faced during design. Nevertheless, the dimension of the problem can be easily reduced through proper evaluation of subsystems and interfaces among them. Once a design objective is identified, it is possible to use optimization techniques to speed up the gathering of information.

For example it is possible to minimize/maximize each stage independently and then moving on to the optimization of the assembly. RSM (response surface methods) can be used to get the design space behavior (i.e. stress distribution under different loads) or compressor maps. Another interesting approach is the statistical one, which allows an appreciation of the impact that one design parameter change can have on the others.
At the beginning of the design activity, identifying the objectives and the interaction between the stages/components of interests becomes relevant in order to establish the correct routine and prioritization of analyses needed.
As when building a house, the foundations can be defined as the design process (see Fig. 1): it is possible to add a brick (functional block) at each step. The various optimizations phase can be connected one another in order to pass information and to refine the entire design. In the compressor stage design, the process identified is useful to decide the parameters that contribute to the important design features: efficiency, reliability, manufacturability, range of functioning and cost.
The workflow also presents connections with other design aspects: the Multidisciplinary block, not expanded in this case, can be integrated with any of the green ones, to any level of detail.
Breaking down the process to the ground level allows the optimization of each component individually, so that when the full machine is put together the optimization can be done on the process and interfaces, eventually taking care of refining the components' design.
In Fig. 2 a workflow is represented dedicated to identifying the best possible diffuser within a stage design (that includes an inlet, an impeller

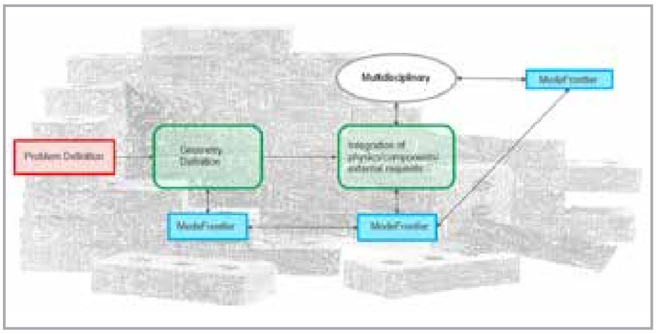


Fig. 1 - Process flow of centrifugal design stage

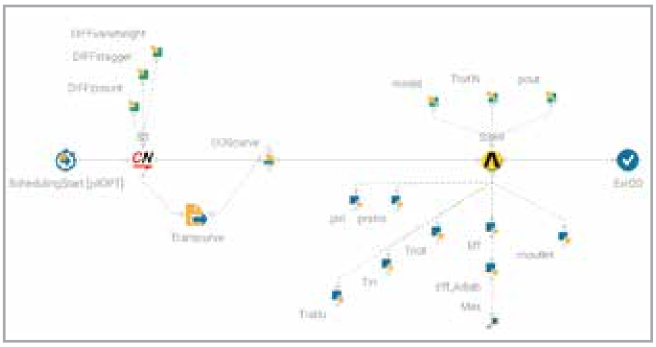


Fig. 2 - Stage workflow

and a diffuser). The design has been segmented to study and define some of the diffuser's parameters. modeFRONTIER is the software used to perform the optimization task. In the software workflow two main nodes are present (where a node identify a software and/or process phase and/or activity). The first node (Concepts NREC) manages design variables and second one (Ansys) is used to run computational fluid dynamics (CFX in the specific). In the first optimization the variable to optimize is just one. It is the first block needed to build the full process.

PilOPT has been chosen as the algorithm to find the optimum even if it is a single variable problem because it is a simple design space with few input variables. This algorithm allows the quick search of the best solution. With these results it is possible to launch a statistical analysis to fully understand the impact of each input variable on the output one. It is then the designer's choice whether to refine the optimization by choosing a Simplex algorithm taking into account statistical results, or to move on to the next design step, maybe integrating the stress evaluation and the definition of the best fillet radii around the vane.

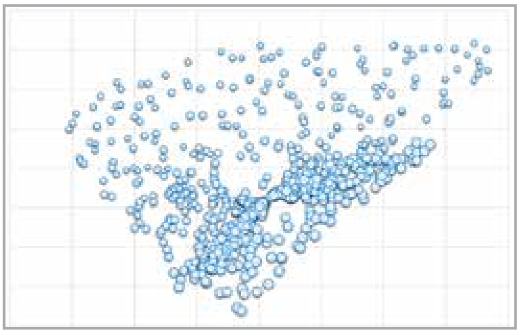


Fig. 3 - results distribution on a pressure-power range

This process can be followed for each and every variable and/or component used to design a multistage compressor. At this point, the structure of the workflow can be the same as Fig. 2, where the geometrical and the Ansys node include multiple stages. The optimization process can follow the same criteria in order to maximize and/or minimize the relevant variables.

In case of a complex workflow or a long computational time, the RSM (Response Surface Method) can improve to speed up the process. Before accessing to this tool, it is fundamental to have a reasonable number of DOE (Design of Experiments) or test cases to use as database. RSM is very powerful but potentially dangerous. It calculates the results based only on the amount of historical data it has, so it is important that this history includes all the areas of the design space (so algorithms which scan the entire design space are preferable for generating the test cases).

For example, a 2 stage-compressor with 1 output and 9 input variables using an algorithm that generates 1665 cases needs 8325 hours of computational time running on 48 CPUs. With RSM and a reasonable number of DOE solutions, the entire space can be scanned and calculated in 2 minutes and 5.5 seconds.

At this point, a judgment has to be made in the results evaluation. Good engineering practice and final design objective guide the selection of results of interest and, if needed, the decision to run the full case. From an entire workflow perspective, RSM helps managing large or complex system, allowing the designer to gather the necessary information in a reduced amount of time, which is valuable from both project and time-to-market perspective.

RSM can have another interesting use, due to its own features. It allows the calculation of a large design space anchored to a good database and can be used to generate maps. It is sufficient to define the problem (correct inputs, output and supporting variables), run DOE or collect test data, then train and run RSM. The result table includes all the necessary information to create maps as happens during a standard CFX simulation. Starting from a complex model the possibility to run in few minutes (i.e. example above) has allowed the generation of a pressure-power map (see Fig. 3), that is used to verify the entire range of operation of the new design.

This methodology can be successfully used to analyze the design space in any context, but it is interesting when physical quantities are among the input variables because it is possible to observe trends for certain phenomena. It is possible to characterize the compressor characteristic curve at different pressure, or inlet conditions, early in design phase in order to change and manage properly the geometrical design variables. For example, the search of stall or surge limits before running a transient analysis of the entire stage can be conducted observing the results of the RSM. The amount of data available allows the selection of critical areas, and supports the possibility to vary some parameters in order to evaluate the impact of changes before final simulation and design assessment are conducted.

Using all the optimization techniques illustrated the designer has the freedom to investigate alternative solutions minimizing the computational cost, including inputs coming from different disciplines (like cost) that typically are known at the end of the design process, and to evaluate a design space broader than what was used to. The tools used in the design process can support the various design goals provide that geometrical and physical quantities are in a parametric format, a process is defined and input/output variables are defined.

About Ingersoll Rand

Ingersoll Rand (NYSE:IR), driven by an entrepreneurial spirit and ownership mindset, is committed to helping make life better. We provide innovative and mission-critical industrial, energy, medical and specialty vehicle products and services across 40+ respected brands designed to excel in even the most complex and harsh conditions where downtime is especially costly. Our employees connect to customers for life by delivering proven expertise, productivity and efficiency improvements.

For more information please visit https://www.ingersollrand.com/

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

Placing computational wind engineering at the fingertips of structural engineers

VENTO AEC benchmarked against reference literature and in a real case

By Marco Mulas
Vento CFD

Computational wind engineering (CWE) is a new science based on computational fluid dynamics (CFD) applied in architecture, engineering and construction. It also simulates a fluid flow that interacts with objects using the discretized Navier-Stokes equations. VENTO AEC is the first CFD software developed exclusively for architecture, engineering, and construction (AEC) professionals, which requires no previous knowledge of CFD and delivers reliable, validated results with unparalleled ease-of-use and speed. This article presents two benchmark cases comparing the results obtained with VENTO AEC to wind tunnel measurements and to the results obtained using Ansys Fluent, as well as a real-world application of this new CWE software package.

Computational wind engineering (CWE) is a fascinating new science that uses computational fluid dynamics (CFD) methods for applications in building and the environment.
From the technical-scientific point of view there are no differences between CWE and CFD. They both simulate a fluid flow that interacts with objects. The discretized Navier-Stokes equations, which govern all fluid flows, cannot distinguish whether the object is a tall building or the model of a car.
The drag coefficient of the car and the pressure coefficient of the building represent the same thing: the wind load.

However, CFD users are very different to CWE users. Mechanical and aeronautical engineers have a much broader knowledge of

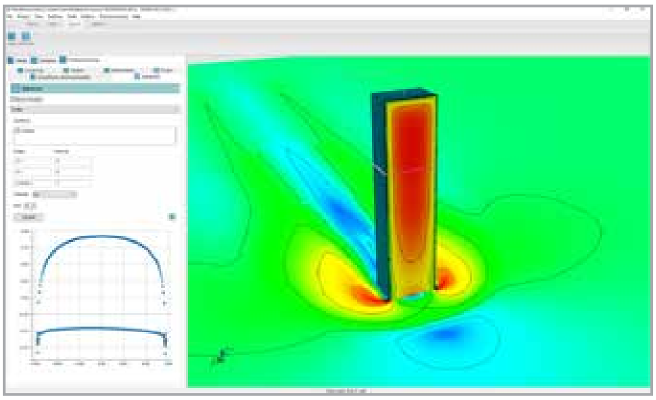


Fig. 1 - VENTO AEC GUI showing the results of a benchmark case (Ref. 1). Color maps of the pressure coefficient Cp on the windward side of the building, and of the air velocity on the ground. Left: Cp distributions on the front and rear side of the building at 2/3 of its height.

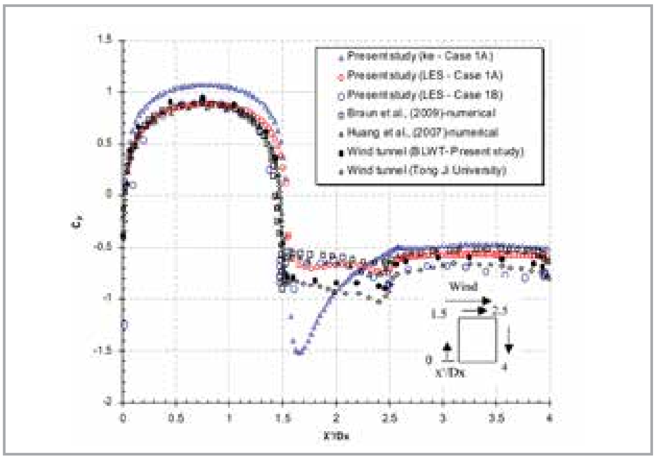


Fig. 2 - Cp distribution in a horizontal section at 2/3 of the height of the building. From left to right: front, side and rear. Results come from two wind tunnel facilities and CFD using various turbulence models. Based on Ref. 1.

fluid mechanics and have been using CFD for 30 years. They are accustomed to navigating the many menus and choosing from a plethora of technical options.

They have mastered the use of the most popular and successful CFD software packages, which are still as complicated, cumbersome, and time-consuming as they were almost 20 years ago. This software is not adapted to the different needs and the environment of civil engineers.

But times have changed. The mathematical methods and physical models used in CFD were mainly developed in the Eighties and Nineties. Nothing noteworthy has appeared in the last two decades in the CFD scientific literature. Analysts and developers, who are familiar with the entire history of its development, are now in a position to potentially simplify the use of CFD software. This is what we have done. VENTO AEC is the first CFD software developed exclusively for architecture, engineering, and construction (AEC) professionals. It was tailored using more than 50 man-years of CFD experience to build a new CWE software that requires no previous knowledge of CFD and delivers reliable, validated results with unparalleled ease-of-use and speed.

The key to this dramatic simplification was a new technology that emerged in the early years of the new millennium, the immersed boundary (IB) technique. In a nutshell, IB technique allows complete automatization of the mesh generation process, which in turn reduces the user's time from many hours or even days, to mere tens of minutes. No other commercial CFD software implements this technique.

In this article, we present two benchmark cases comparing the results obtained with VENTO AEC to wind tunnel measurements and to the results obtained using Ansys Fluent, the most popular CFD software on the market since the Eighties.

The first case involved the determination of the pressure coefficient (Cp) on a model of a tall, isolated building [1]. The model is 42cm high. Fig. 1 shows the results from VENTO AEC, specifically the Cp color map on the windward side of the building, and the color map of the wind speed on the ground. On the left, the user can extract the Cp distribution on both the windward and the leeward sides in an arbitrary section of the building that can be seen in the same image.

Fig. 2, taken from the reference literature, shows the Cp distribution obtained in two different wind tunnel experiments, and from a variety of Ansys Fluent numerical options. It also shows the Cp distribution on the lateral side of the building, between the front and the rear.

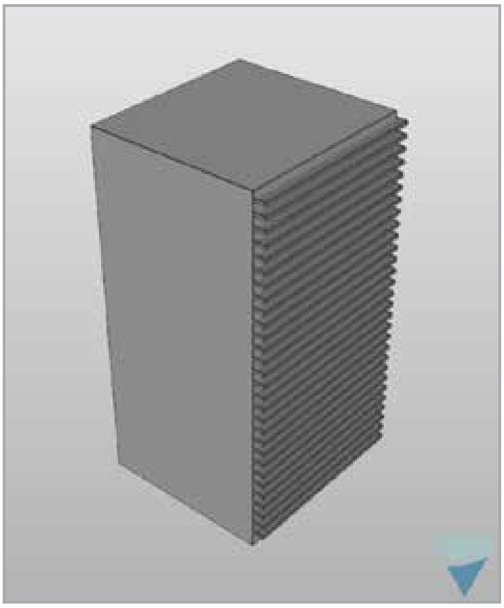


Fig. 3 - Benchmark no. 2: model of the 30-story building with balconies (Ref. 2).

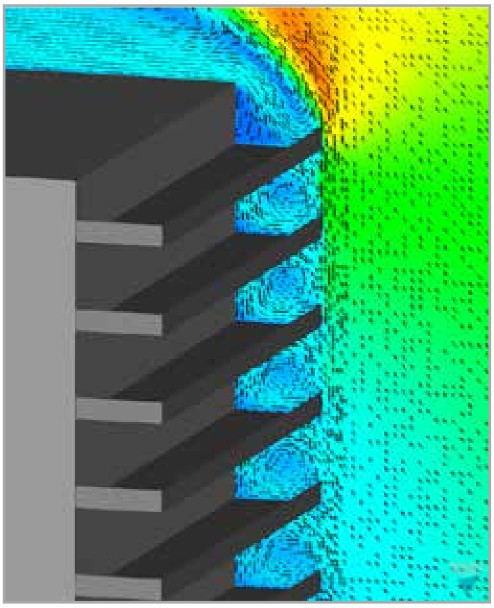


Fig. 4 - Benchmark no. 2: color map of the air velocity in a vertical section on the balconies

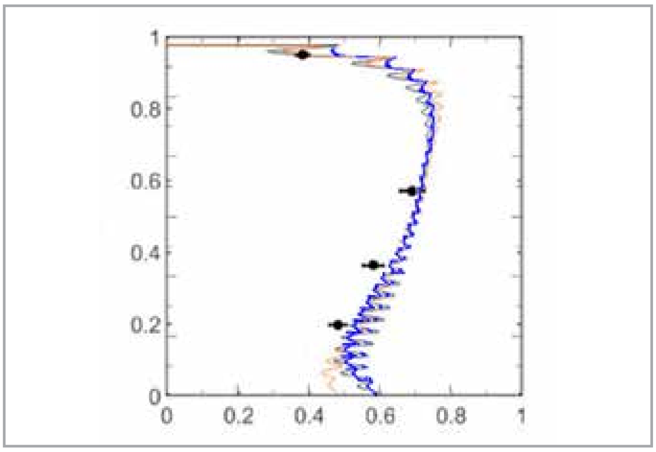


Fig. 5 - Benchmark no.2: vertical Cp distribution on the section shown in Fig. 4. Blue line: VENTO AEC. Orange line: Ansys Fluent RANS model. Grey line: Ansys Fluent LES model. Black symbols: experimental measurements.

It is interesting to note that all but one of these results match almost perfectly on the windward side. The various distributions on the leeward side are distributed in the range -0.5 to -0.7.

The second test case is similar to the first, but more recent. In this case, the 30-story model of a building has balconies on the windward side [2]. Fig. 3 shows the building model. Fig. 4 shows the color map of the wind velocity in the balconies for a vertical section. The more interesting image is seen in Fig. 5 which shows the vertical Cp distributions for the same vertical section as Fig. 4.

In Fig. 5, the VENTO AEC results are displayed in blue and the Ansys Fluent results are displayed in orange (RANS model) and gray (LES model). The black symbols are measurements taken by four experimental probes. The LES model is a far more sophisticated CFD modelling technique for simulating turbulence. It is considerably more time-consuming but also more accurate that the RANS model.

RANS is the standard turbulence modelling technique used by CFD software, and also by VENTO AEC. The numerical results of the three approaches are essentially superimposed for most of the height of the building. It is interesting to note the behavior of VENTO AEC near the ground, where its output corresponds to Ansys Fluent's more accurate LES results.

Finally, we want to present a real case. In recent months, we have collaborated with Fondazione Fiera di Milano, and the engineering firm Studio Marzullo in Rome (which was responsible for the structural design of Fiera Milano), to determine the wind load on the pavilions of Fiera Milano, the Milan international exhibition center.

The client planned to place more than a thousand photovoltaic panels on the roof of the pavilions (measuring approximately 35-thousand square meters each), and it was decided to rely on a wind analysis instead of on the "Normative" (the Italian code for the evaluation of wind action on buildings).

Fig. 6 - Fiera Milano. a) A view from Google Earth; b) The 3D model.

CWE permits a realistic geometrical representation compared to the necessarily highly approximated regulations. It also allows you to include all the surroundings in the model. Figs. 6a and 6b show a photo taken by Google Earth and the 3D model used for the numerical analysis, respectively. The mesh consisted of almost 20 million cells.

One of the most interesting parts of our investigation was focused on a pavilion with a pitched roof of about one degree. We compared the wind actions using three different models: the complete model shown above in Fig. 6b; the actual stand-alone model of this pavilion (without any nearby buildings); and a rough geometry with no beams or pillars, and with a flat roof free of panels. The last model was intended to be a close approximation of the geometry used in the code.

The following table shows the vertical upward force on the roof and on the panels. The wind speed was 90km/h (25m/s):

Model	Force on the roof [tons]	Force on the panels [tons]
Complete model of the exhibition center	170	6
1 pavilion only	300	19
Simplified pavilion	370	0

The table shows the effects that the use of a realistic model has on load determination. Note that the difference between models 2 and 3 was essentially due to the actual shape of the pavilion which included pillars at the four corners and a cantilevered beam of about 6 meters.

There are two reasons for the significant decrease in wind load obtained using the complete model. The first and less important was the presence of a small urban district upwind of the pavilion that slowed down the wind as it approached the pavilion. The second, more important cause was the presence of the small building adjacent to the pavilion on the windward side. That building absorbed most of the wind load (minimum Cp), reducing the pressure on the pavilion.

Figs. 7a, 7b and 7c show the two color maps of the wind speed near the ground and the Cp distribution on the roof. With reference to the Cp map, the red color corresponds to Cp=0, so it is the neutral area that does not contribute to the force. The wind direction was orthogonal to the short side in the foreground.

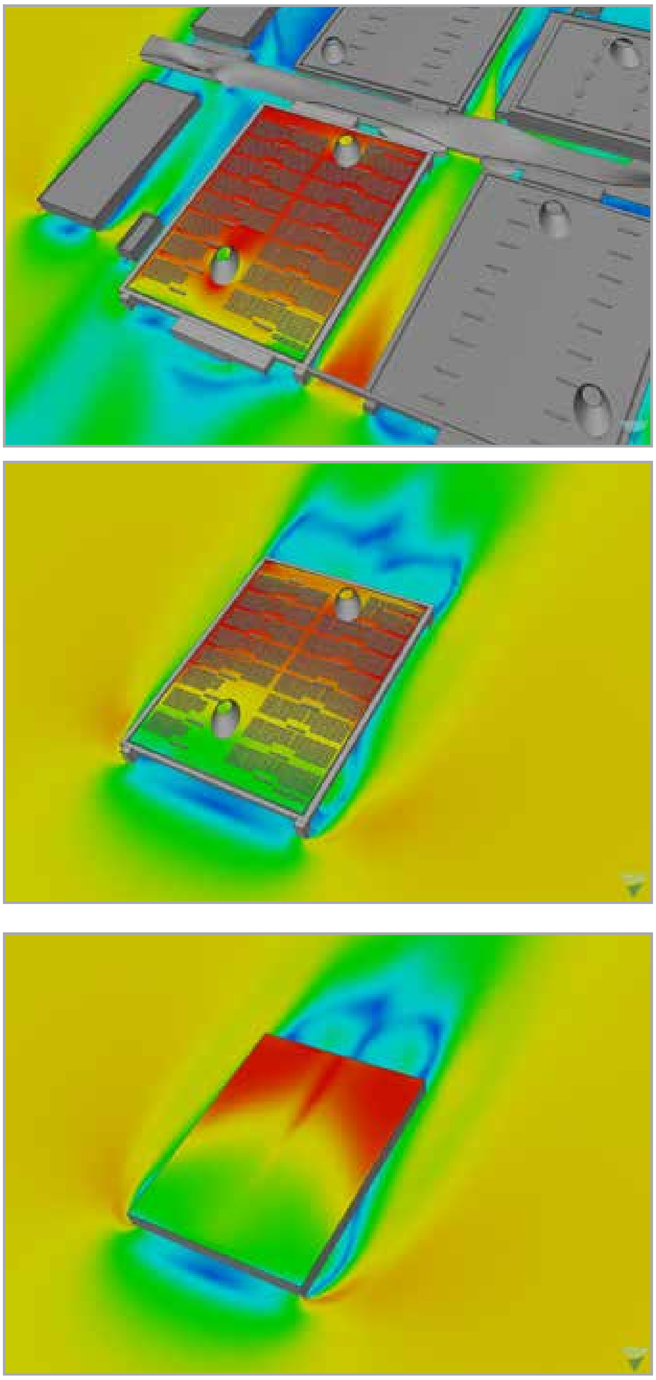


Fig. 7 - Color maps of the pressure coefficient Cp on the roof of the pavilion, and of the air velocity near the ground: a) The complete model; b) The stand-alone pavilion model; c) A simplified model of the pavilion.

References

[1] A.K. Dagnew et al. "Computational evaluation of wind pressure on tall buildings", 11th Americas Conference on Wind Engineering, San Juan, Puerto Rico, June 22-26, 2009

[2] X. Zheng et al. "CFD simulation of wind flow and mean surface pressure for buildings with balconies: comparison of RANS and LES", Building and Environment 173 (2020) 106747

Fig. 8 - The photovoltaic panels being mounted on Fiera Milano's pavilions.

The last image we are pleased to present is the one shown in Fig. 8. This is a picture taken a short time ago showing the photovoltaic panels being mounted on Fiera Milano's pavilions.

Acknowledgements

We would like to thank Fondazione Fiera di Milano and Studio Marzullo for their permission to show the images of the analysis.

For more information:
Marco Mulas - Vento CFD
info@vento-cfd.com

How to stay operational in a virtual world

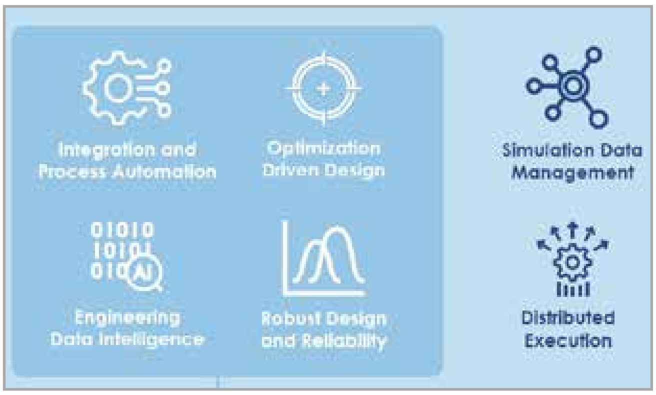
By Novella Saccenti
EnginSoft

Introduction

Designing, developing and manufacturing high-performance, ground-breaking products requires engineers to integrate multiple disciplinary skills into an effective and efficient process of design and optimization.

Furthermore, the need to bring to market Industry 4.0-ready solutions at competitive production and operational costs, demands a systemic vision capable of implementing virtual prototypes that not only emulate functional and physical behaviors, but also support decision-making in all the product's life-cycle scenarios.

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



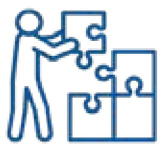
The conceptual and pre-design phases are then organized in the best way, having as focal point the vision of the whole design process and its connections with singular disciplines. People involved in each discipline are outlined and assigned to specific activities, avoiding overlapping of work areas. In the context of spreading knowledge, all the results are stored in a shared environment, making them available for all the allowed users.
The Time to Market needs to be improved constantly, reducing costs derived from huge numbers of numerical simulations and physical prototypes. In this perspective the automation of processes represents the essential step to move on to the Industry 4.0 environment, making it possible to reduce the number of hour/man dedicated to actions that could be made by artificial intelligence.

These goals can be easily achieved, using platforms expressly created for managing the amount of data generated by a company.

What is a Simulation Process and Data Management (SPDM)

A Simulation Process and Data Management is a technology able to organise data in a shared environment. Nowadays companies need the effective management of data coming from various sources, such as numerical simulations, experimental data, financial analysis, etc… It is because of this necessity that technologies such as a SPDM are being used more and more.

Sharing and knowledge of data



Technicians can be used to work alone, in a local environment and take advantage of cloud and shared folders to communicate with their colleagues. When many different departments have to collaborate, issues related to the sharing of data can arise: that's why the creation of a shared environment is one of the focal point of an SPDM system.

A company can be the best in its field due to the knowledge of its people: in this perspective, the loss of a technician can be the loss of an important slice of knowledge for the whole company. The SPDM gives the possibility to record and to maintain the knowledge inside the company, having tracked and stored all the essential files, reports and analysis. This work can also be done with a basic database, but the power of a SPDM is that data are organized in a smart way: assigned labels and metadata structures allow an advanced search for what is needed, in the smaller amount of time.. In this way, it is possible to choose the most profitable solutions and shorten product-development cycles.

Processes do not only need to be shared and stored, but it is essential to have them automate: transferring data from one discipline to another requires a lot of time and high probability of errors. Automatizing the process brings the advantage to make the artificial intelligence to the dirty job, with no errors, leaving higher value actions to the human.

Versioning

Simulating is not a static process: it is made of lots of different versions of the same basic design. How can we track all the changes, who made them, when and what has been changed? The versioning feature becomes then the answer: tracking the versions of the same model builds the history of that model, giving information about its life, from the start to the end. From this perspective, the capability to recover an old design is immediate, since it is possible to address the right version.

Users education

The technical features are given by the technology, but they are nothing without the correct education of users: labels, versions and metadata are suggested from the platform and must be, at some percentage, compiled by a human.

The company must then give a clear message to the departments, regarding the correct behaviour of users, in order to properly exploit the capabilities of the SPDM. Some indications must be outline:

- All users must agree to the change in the working modus, given by the SPDM: process are mapped and completely clear from the beginning.
- Activities must be assigned to specific users, defining their limits.
- Rules about naming, creation of folders, saving of simulations and the general deployment environment must be sketch and defined at the beginning of the project.

One important factor to be considered is that targets belonging to single disciplines may not be aligned with the broader business strategy: this is the reason why awareness of the multidisciplinary framework must be taken into consideration.

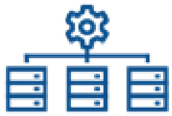
Effects on hardware and company technologies

The usage of the SPDM is not only influencing people who will work with it, but also the hardware needed to support the technology must be up to date. It is then fundamental that the IT department knows exactly the development and the effect the SPDM will have on the hardware. In this context the automation of processes pays a major rule: the link with software and technologies already installed must be analysed to make the connection effective. As an example, all the medium/big companies have platforms like PLM, which are essential tools to be used. The SPDM must interact with the PLM creating a bridge towards the idea of an all-in-one collaborative way of working. The elements of the integration vary from CAD, materials, product performances and configurations.

Simulations are nothing without the comprehension of results: the post processing is then an essential part of a technology. It needs to be enough rich of charts and statistics to make the user able to know the behaviour of data and then make the best decision.

Automation of processes also lies in the automatic creation of reports, enabling one fixed layout, easy to understand and shared across departments.

Security of data



Security and privacy of data are nowadays an essential factor when a new technology is been adopted. For this reason, the management of security barriers is widely developed in the SPDM. Storage, access and viewing of data are accessible only by users with permissions, addressed by the administrator of the platform. Permissions can be changed from project to project in order to replicate real situations.

Conclusions

The Simulation Process and Data Management technology is the new up-to-date development platform for companies that want to be part of the Industry 4.0 revolution.
Its features encounter contemporary needs for the new pillars of technology: shared knowledge, security of data, smart exploitation of hardware resources and automation of processes.

The objective is to cover all activities in order to organize and manage engineering workflows and support the management of data even as input for or results of computational engineering tasks and tools. During these times of remote work it is fundamental to understand the importance of technologies able to preserve the productivity of a company, without loosing time and resources.

For more information:
Silvia Dematté - EnginSoft
s.dematte@enginsoft.com

Increasing the ability of aircraft designers to predict aircraft performance changes due to ice accretion

De-icing analysis of an electric heater mat using Ansys FENSAP-ICE

By Jae Seung Choi
TAE SUNG S&E

During flight, as certain altitude and temperature conditions reach freezing point, supercooled water droplets in the clouds accumulate on the surface of the aircraft and freeze.
Aircraft ice accretion mainly occurs on the wings, which changes the airfoil shape, decreases the lift force and increases the drag force. This change in the aerodynamic characteristics reduces the overall performance of the aircraft. If ice accretes on the control surface of the aircraft, at worst, the control surface will no longer function. In addition, if ice accretes at the engine inlet, pressure loss will increase and engine performance will decrease. Ice may flow into the engine compressor and cause the engine to malfunction, causing a serious aircraft crash.
In this study, we removed and analyzed the ice after it was produced in a helicopter engine inlet equipped with an electric heater mat, which is a de-icing system to remove the ice accretion accumulated in the main part of the aircraft.

In-flight icing

In-flight icing is caused when supercooled water droplets still in liquid form at temperatures below freezing point, are encountered in a cloud. When the aircraft enters an icing cloud, these supercooled droplets hit the front area of the aircraft surface, forming an icy layer whose roughness and shape can create distortions in the aerodynamic performance of the wings, propellers, etc.

Loss of aerodynamic performance (lift, drag and momentum) and deterioration in stability and control as a result of freezing occur due to:

- Lower lift at a given angle of attack (AoA)
- Reduced maximum lift coefficient
- Reduced stall margin (lower AoA up to stall and higher speed to avoid stall)
- Increased drag due to roughness and flow separation
- Increased weight and shifted center of gravity
- Stability and control problems from asymmetrical roughness

Fig. 1 - In-flight aircraft icing

Icing usually blocks the engine intake and internal ducts, increasing pressure loss and, if ingested, can damage components inside the engine and cause loss of thrust and engine malfunction. More specifically, an IPS (icing protection system) installed at an engine intake is divided into two systems: an anti-icing system and a de-icing system. The anti-icing system uses electricity to maintain a temperature at which freezing does not occur, while a de-icing system raises the temperature to some extent after freezing to reverse freezing. An electric heater mat is based on the second type of IPS (icing protection system) and this article will describe an example of de-icing analysis at the inlet of a helicopter engine.

Ansys FENSAP-ICE

Ansys FENSAP-ICE was developed as an aircraft icing analysis program and consists of the following four modules (Fig. 2):

- the FENSAP module analyzes dry airflow;
- the DROP3D module analyzes wet airflow and the water droplet field;
- the ICE3D module analyzes the ice accretion; and
- the CHT3D module analyzes the conjugated heat transfer.

All four of these modules are applied to perform a de-icing analysis.

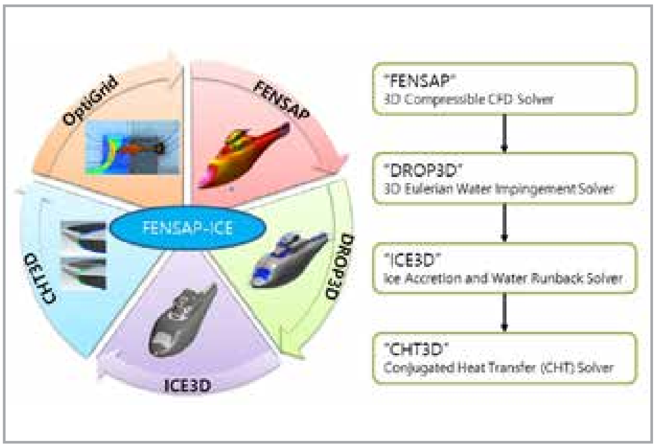


Fig. 2 – Ansys FENSAP-ICE schematic

Example of de-icing analysis

The configuration for analysis (see Fig. 3) is a genetic helicopter with two engine inlets (on the left and right sides) each with an inlet diameter of 0.2m. The electric heater mat mounted on the surface of the engine inlet is stacked in three layers, with the heating wire installed in the inner layer of insulation material, while the outer layers are composed of fiberglass to prevent corrosion. The properties of each material are shown in the table below.

The mesh (see Fig. 4) to conduct the conjugated heat transfer analysis consisted of a fluid zone including the helicopter fuselage and engine inlet, and a solid zone representing the electric heater mat, for a total mesh count of approximately 16 million cells.

For the dry airflow analysis, level flight conditions were applied at a flight speed of 70m/s (altitude=0.0ft / temperature=-10°C); while for the wet airflow and droplet field analysis liquid water content (LWC)=1.0g/m3, and mean volume diameter (MVD)=20μm were applied. The engine flow rate was 2kg/sec, and the wall temperature for calculating the wall heat flux reflected 2.4°C, which is +10°C more than the total temperature.

For the de-icing analysis of the electric heater mat, an equal heating source of 5.1kW was applied to each side, with a heating cycle that operated for 30 seconds and was idle for 30 seconds, for a total of 600 seconds (see Fig. 5).

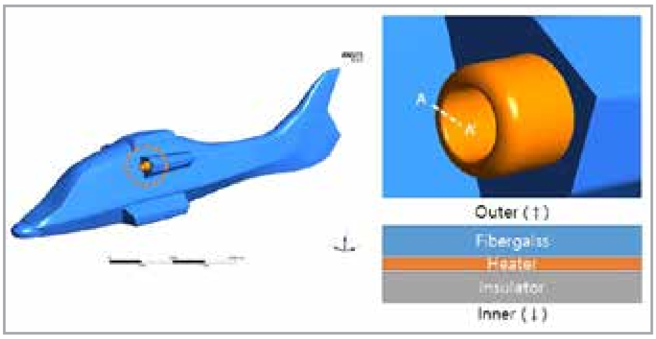


Fig. 3 – Configuration of genetic helicopter and electric heater mat

Layer	Material	Material Property		
		Density (kg/m^3)	Thermal Conductivity (W/m/K)	Enthalpy at 0°C (J/kg)
Outer	Fiberglass (1.0 mm)	2,700	0.313	393,610
Center	Heater (0.2 mm)	4,540	17.03	141,310
Inner	Insulator (1.0 mm)	89	0.025	5,267,059

Table 1 – Material properties of the electric heater mat

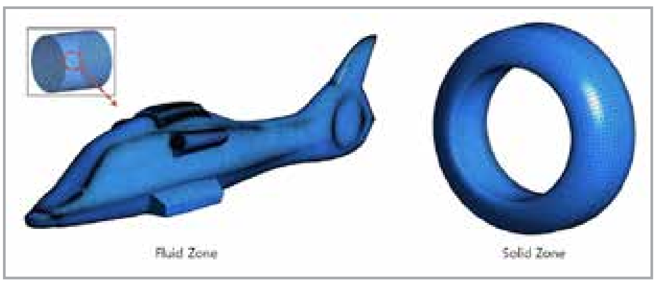


Fig. 4 – The mesh consisted of a fluid zone and a solid zone

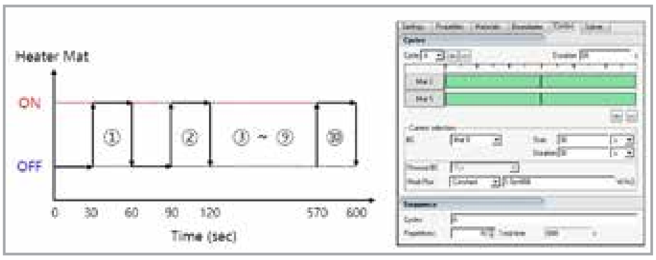


Fig. 5 – Operating conditions of the electric heater mat

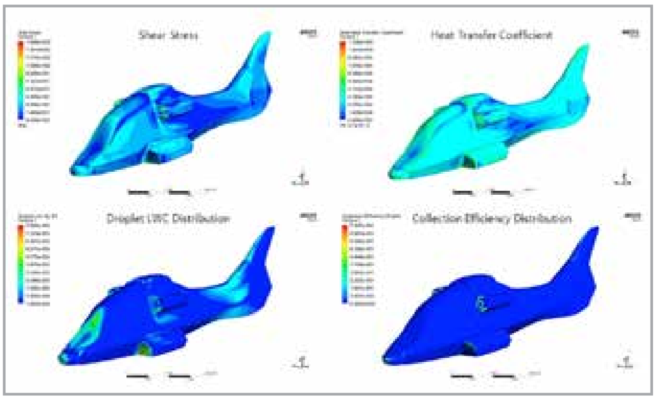


Fig. 6 – Dry airflow and wet airflow analysis results

The results from the dry airflow are the shear stress and the surface heat transfer coefficient values. These results are applied in the wet airflow and droplet field analysis to calculate the collection efficiency (β), also called water catch efficiency.

The above analysis results are used in the CHT3D module for the heat transfer analysis after which the FENSAP-ICE workflow, as shown in Fig. 7 below, is conducted. Heat flux (W/m2) and surface temperature (K) are iteratively calculated in the interface between the fluid zone and the solid zone. Unsteady de-icing analysis with specific time step (0.1sec) is conducted during total icing time (600sec) in this way.

The temperature variation over time on the surface of the electric heater mat is as shown in Fig. 8 below. The temperature increases and decreases periodically according to the heating cycle. Ice accretion is removed in the zone above freezing point, while ice accretes in the zone below freezing point.

Fig. 9 below shows the change in the surface temperature of the electric heater mat as it increases and decreases periodically, with the resultant deicing and icing on the inlet surface accordingly.

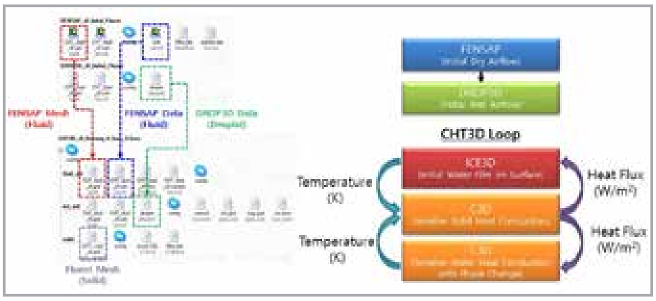


Fig. 7 – Unsteady ice accretion and melting in the loop

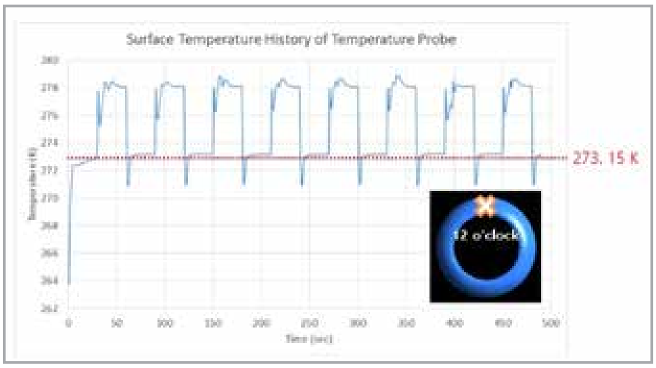


Fig. 8 – Time history of the electric heater mat's surface temperature

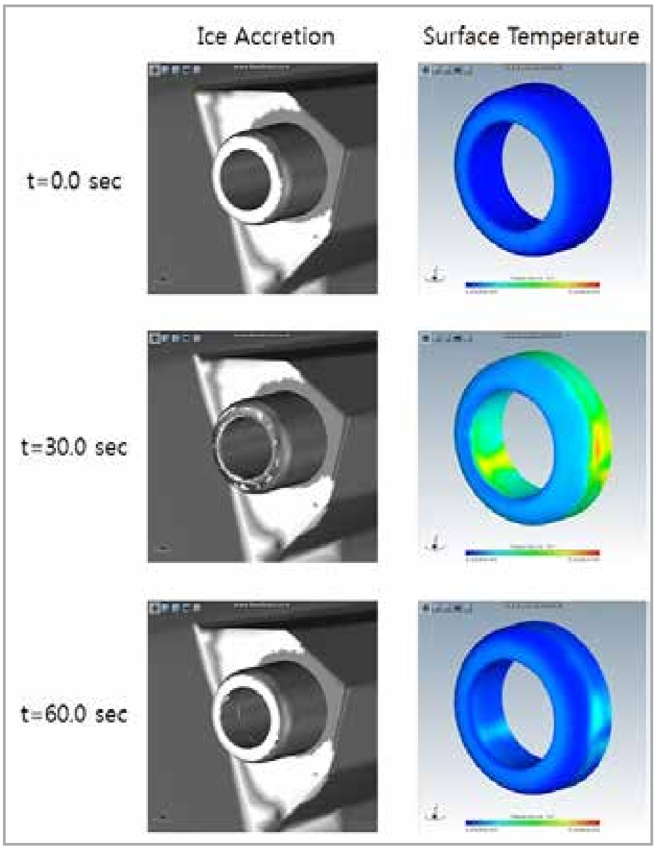


Fig. 8 – Time history of the electric heater mat's surface temperature

Conclusion

In-flight aircraft icing inevitably occurs when certain icing conditions are satisfied, and it can affect the performance and safety of an aircraft. To eliminate this risk, the helicopter studied is having an anti-icing/de-icing system installed.

As demonstrated in this study, Ansys FENSAP-ICE can conduct a de-icing analysis on the performance of the de-icing system using a basic icing accretion analysis. We believe it can be useful to designers for predicting performance changes from alterations to various design variables, such as the area of installation of the IPS(icing protection system), the values of the heating source, and the heating cycle time.

For more information:
Jae Seung Choi - TAE SUNG S&E
jschoi@tsne.co.kr

Testing a mathematical model to determine the destructive rotational speeds and the residual elongations of gas turbine engines (GTE)

Using Fidesys to simulate the development of large plastic deformations

By A.N.Servetnik
Head of the Department of Dynamics and Strength of Aircraft Engines at the Central Institute of Aviation Motors (CIAM) in Russia

Following approval of the software and hardware system based on CAE Fidesys, the problem of modeling the development of large plastic deformations in a rotating model disc for a gas turbine engine was solved. The results of the mathematical model calculations using physical and geometric nonlinearity models showed excellent convergence with experimental data, demonstrating the validity of replacing some of the tests with mathematical models to reduce testing time and costs.

Determining the destructive rotational speeds and the residual elongations of gas turbine engine (GTE) discs is a mandatory requirement of their design and certification. For this purpose, acceleration tests of discs or rotors are usually conducted in special installations [2]. Based on its numerous years of experience, the team at the Russian Central Institute of Aviation Motors (CIAM) understands that substituting some of these tests calculations based on mathematical models is important to reduce the testing time and costs.

In recent years, with the active development of numerical methods, in particular the finite element method (FEM), it has become possible to investigate the bearing capacity of discs of complex configuration, and to use 3D calculation schemes together with plastic flow theory[3,4]. The main idea of modeling is an almost static, step-by-step calculation of the disc with an increase in the external load on the design model. For most structural alloys used in the manufacture of GTE discs, this process is accompanied by the development of large plastic deformations up to destruction.

In this study, we simulate the process of spinning up a model disc under the influence of a gradually increasing centrifugal load,

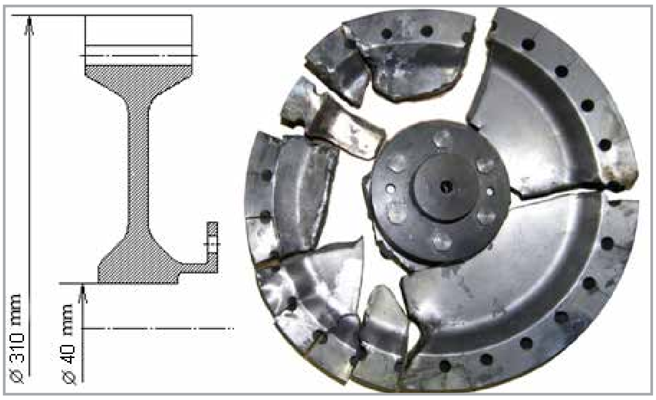


Fig. 1 – The sketch of a model disc (a) and a destroyed model disc (b)

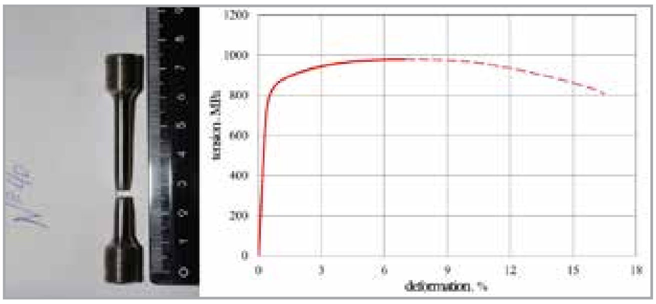


Fig. 2 – The results of the tensile tests of the sample
(a) The sample after rupture (b) The tension diagram

taking into account physical and geometric nonlinearities using CAE Fidesys, an industrial system for strength engineering analysis. To check the model's performance, the calculated elongations of the outer diameter of the model disc were compared with experimental data.

Research objects and full-scale testing of samples

The subjects of the research were two blanks of a disc cut from a 330mm-diameter steel rod. A model disc for the acceleration tests was made from the first blank (Fig.1a) and ten cylindrical specimens for the tensile tests to construct the curing function were made from the second blank.

The disc acceleration tests were conducted at room temperature in one step until the fracture into fragments. During the test, the rotation speed and the elongation of the outer diameter of the disc were recorded according to the method described in [2, 5]. The object collapsed at a rotation speed of 24,282rpm, while the elongation of the outer diameter of the disc at the time of destruction was 4.4mm. Figure 1b) shows a destroyed model disc. Initial fracture occurred in a cylindrical section with a radius R = 55mm.

Estimation of the bearing capacity of the disc according to the theory of limit equilibrium [6] has shown that the destruction of the disc occurs in a cylindrical section with a radius R = 56mm at a rotation speed of 23,100rpm, which satisfactorily coincides with the test results.

The tensile tests of the samples were conducted at normal temperature with continuous recording of the deformation diagram up to the beginning of the formation of a "neck" in the sample (Fig. 2a). The conditional deformation curve plotted from the average values of

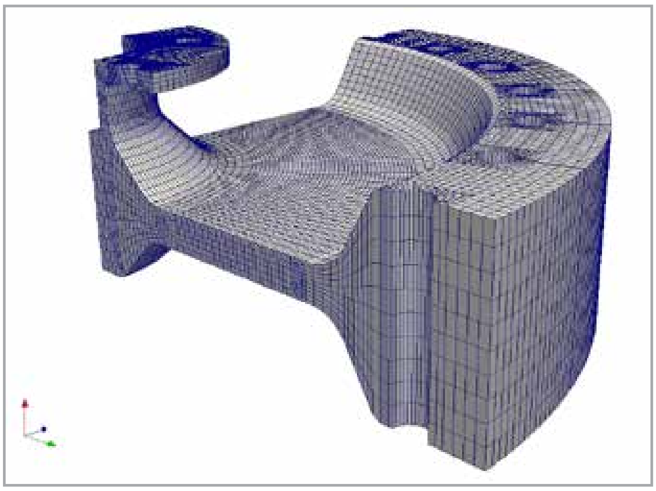


Fig. 3 – FE mesh of the model disc

the mechanical characteristics is shown in Fig. 2b. The dashed line shows the section of the tension diagram after the neck formation began.

Calculation model for computer simulation of tests

The computational model of the disc is a sector with a 90° angle at the base (Fig. 3). Along the meridional sections of the sector, the boundary condition is set as equal to zero of the circumferential displacements.

A centrifugal force was applied to each node of the computational model, proportionate to the square of the rotation speed and the current radius. To generate the hardening function, the conditional deformation curve was reconstructed like the true one according to an assumption of a constant volume of the working part of the sample before and after plastic deformation [7].

Calculation results

One of the phenomena characteristic of field experiments that occurs immediately before the destruction of the disc is a localized "tightening" of the disc. This is similar to the "neck" in the sample and is characterized by the development of nonlinear elastoplastic instability.

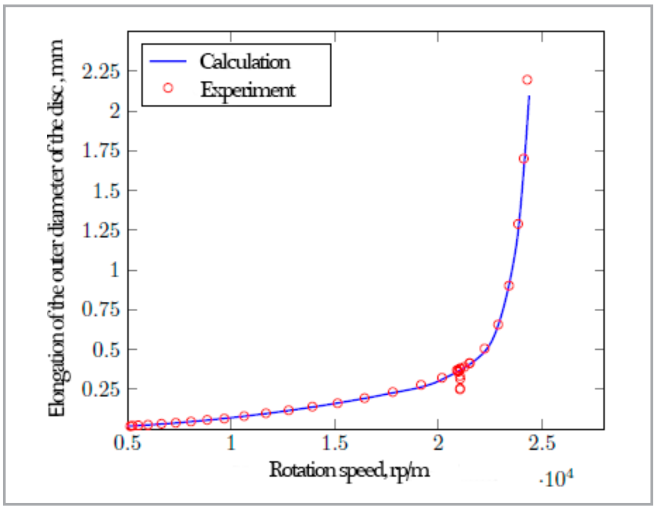


Fig. 4 – Calculated and experimental curves of the elongation of the outer diameter of the model disc, according to the rotation frequency

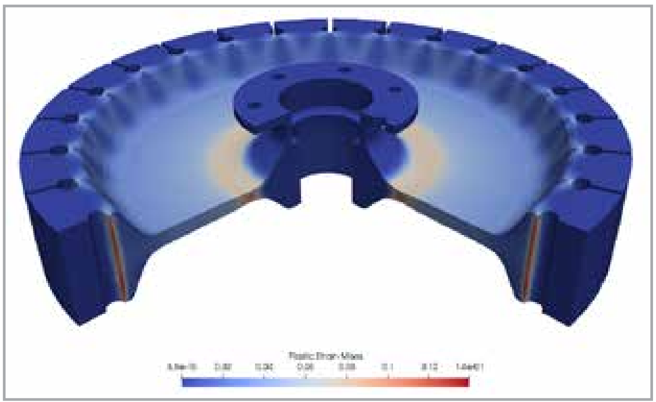


Fig. 5 – The distribution of plastic deformations in the disc at rotation frequency ω = 24,375rpm (dimension to scale - relative units)

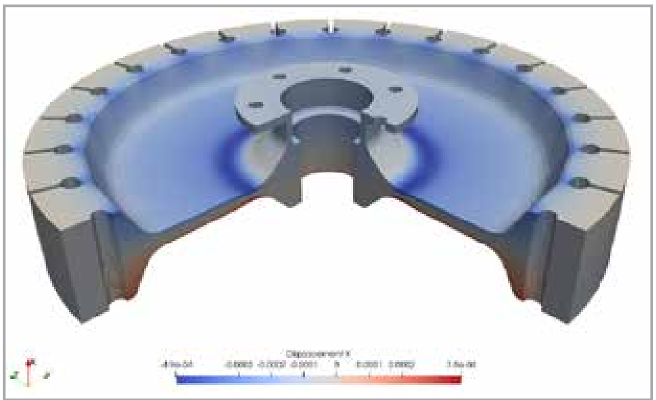


Fig. 6 – Distribution of the axial displacements in the model disc at rotation frequency ω = 24,375rpm (dimension to the scale - m)

To simulate this effect, a large number of loading steps (with a gradually increasing disc rotation speed) and iterations of a nonlinear algorithm (Newton-Raphson method) were performed at each step. At the same time, due to the high nonlinearity of the problem, it was necessary to use a direct solver for systems of linear algebraic equations, which are the most demanding on computational resources.

The results of the calculations of the elongation of the outer diameter of the model disc in CAE Fidesys and their comparison with experimental data are presented in Fig. 4. We see that the curves are in excellent agreement with each other. Beginning with a rotation frequency of 20,000rpm, we observe a significant change in the growth rate of the model disc elongations, with a dependence close to vertical as the destructive rotation frequency is approached. At a rotation frequency ω = 24,375rpm, close to the fracture point, the maximum plastic deformations are located at the disc radius R = 50-60mm with the formation of a “tightening” (Figs. 5 and 6), consistent with the fracture zone of the model disc.

About Fidesys

The CAE Fidesys software package for high-precision strength calculations is the main product of the Russian company FIDESIS. The software package is available in traditional desktop [13] and cloud versions [14]. The project began in 2009 at the Moscow State Lomonosov University under the guidance of Professor Vladimir Levin [8-12]. The company then became a resident of the Russian Skolkovo Foundation, which was the impetus for the creation of the first commercial version of the product, and subsequently the cloud version of CAE. Today we employ 25 mathematicians, graduates of leading specialized universities and PhDs. In addition, the organization employs 11 consulting professors from Moscow State University, MEPhI, Moscow Institute of Physics and Technology (MIPT), scientific institutions of the Russian Academy of Sciences, the University of New Hampshire, and Columbia University. FIDESIS provides engineering consulting and custom software development services to solve highly specialized tasks for its customers.
The company is involved in: complex problems of mechanical resistance and stability of structures; geophysics and geomechanics; strength of composites; intelligent materials; solid phase transitions.

References

[1] Nozhnitsky Yu.A. The Problem of Ensuring Reliability of Gas Turbine Engines. ATCES 2017, IOP Conf. Series: Materials Science and Engineering, 2018, no. 302 (2018), 9 p. DOI: 10.10.1088/1757-899X/302/1/01/20182..

[2] Yu.A. Nozhnitsky, B.A. Baluev, Yu.A. Fedina. Eksperimentalnye issledovaniya prochnostnoy nadezhnosti perspektivnyh gazoturbinnyh dvigateley [Experimental studies of the strength reliability of promising gas turbine engines]. Vestnik UGATU, 2015, vol. 19, no. 3(69), pp. 4-14.

[3] Servetnik A. Energy-based method for gas turbine engine disks burst speed calculation. 28th Congress of the International Council of the Aeronautical Sciences (ICAS2012), Brisbane, 2012

[4] Nozhnitsky Yu, Karimbaev K., Servetnik A. Numerical simulation of spin testing for turbo machine disks using energy-based fracture criteria. ASME TURBO EXPO, Copenhagen, 2012.

[5] Nozhnitsky Y.A., Servetnik A.N. Prevention of hazardous failure of the turbine rotor due to its overspeed // IOP Conf. Series: Materials Science and Engineering. – 2018. – No. 449. – 10 p. DOI: 10.1088/1757-899X/449/1/012025

[6] Shorr, Boris. (2015). Creep and Durability at Non-stationary Loading and Heating. 10.1007/978-3-662-46968-2_8..

[7] Shorr, Boris. (2015). Unsteady Thermal Plasticity. 10.1007/978-3-662-46968-2_5.

[8] A. V. Vershinin, V. A. Levin, A. V. Kukushkin, and D. A. Konovalov. Structural analysis of assemblies using non-conformal spectral element method. IOP Conference Series, 747:012033, 2020.

[9] Dmitriy Konovalov, Anatoly Vershinin, Konstantin Zingerman, and Vladimir Levin. The implementation of spectral element method in a cae system for the solution of elasticity problems on hybrid curvilinear meshes. Modelling and Simulation in Engineering, 2017:1–7, 2017.

[10] V. S. Karpenko, A. V. Vershinin, V. A. Levin, and K. M. Zingerman. Some results of mesh convergence estimation for the spectral element method of different orders in fidesys industrial package. IOP Conference Series: Materials Science and Engineering, 158:012049, 2016.

[11] G. P. Nikishkov, A. V. Vershinin, and Y. G. Nikishkov. Mesh-independent equivalent domain integral method for j-integral evaluation. Advances in Engineering Software, 100:308–318, 2016

[12] V. A. Levin. Theory of repeated superposition of large deformations: elastic and viscoelastic bodies. International Journal of Solids and Structures, 35:2585–2600, 1998.

[13] www.cae-fidesys.com

[14] www.sim4design.com

For more information:
Anatoly Vershinin - Fidesys LCC
a.v.vershinin@cae-fidesys.com

Treating residual stresses and deformations in die castings

Virtual simulation during product and process design improves time to market and cost efficiency

By Giampietro Scarpa, Francesca Lago
EnginSoft

This technical article illustrates the advantages of using virtual simulation to evaluate the thermal processes of die casting to reduce the formation of residual stresses and deformations in the components produced. Simulation is becoming increasingly valuable in this regard as the geometrical complexity of castings increases to meet market demand for more competitive, higher quality products at lower cost and reduced time to market.

Growing demand for higher and higher quality die castings with increasingly complex geometrical shapes requires the components to be designed together with their production processes in order to minimize time, cost and waste on the one hand, and maximize mechanical performance on the other.

Product design is, in fact, evolving towards increasingly intricate and lightweight components. In addition, the evolution of electronics in the automotive field is drastically boosting the performance of larger frame components characterized by reduced wall thicknesses, complex shapes, and exceedingly small tolerances.

However, these factors also affect the development of residual stresses and deformations during the production process, which can influence proper assembly of the parts and component performance during use.

Simulation serves as a useful tool to predict all the component's qualitative information, including the stresses and deformations occurring at the end of the process.

Stresses and deformation can form in different phases of the melting process, specifically: during the solidification of the component inside the mold; during the ejection of the casting from the mold; during the cooling phase after extraction; during blanking; or following any heat treatment.

In die casting, one of the main causes of residual stresses and deformations typically arises from the fact that the component does not cool simultaneously at every point, with the outer surface cooling first followed by the core of the material and the thin-walled areas, before the more solid parts. There are also the bonds with the mold, for instance to prevent part ejection, during the solidification and cooling phase that may induce particularly high contact pressures in some areas of the component.

Given this tight contact between the part and the mold, ejection by the extractors, if badly positioned, may cause deformations in the component because the alloy has a low mechanical resistance while it is still at a fairly high temperature (about 300 °C- 350 °C), see Fig.1.

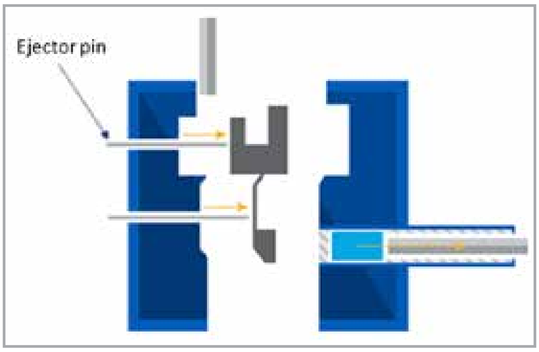


Fig.1 – Extraction from the mold

Residual stresses and deformations can be corrected with manual techniques later in the production process, for instance, by straightening with jigs, or with dimensioning by machining specific allowances; alternatively, virtual simulation allows you to predict these problems in advance and identify the appropriate corrective actions to be taken.

Starting with the simulation of the thermal treatment, which mimics the production reality, you can study the component's thermal evolution through the various phases of the production cycle from injection and solidification, to the opening of the mold with the lubrication and blowing steps, while taking into account the effect of the thermoregulation circuits up to extraction and subsequent cooling in water, as shown in the case study below (Fig.2).

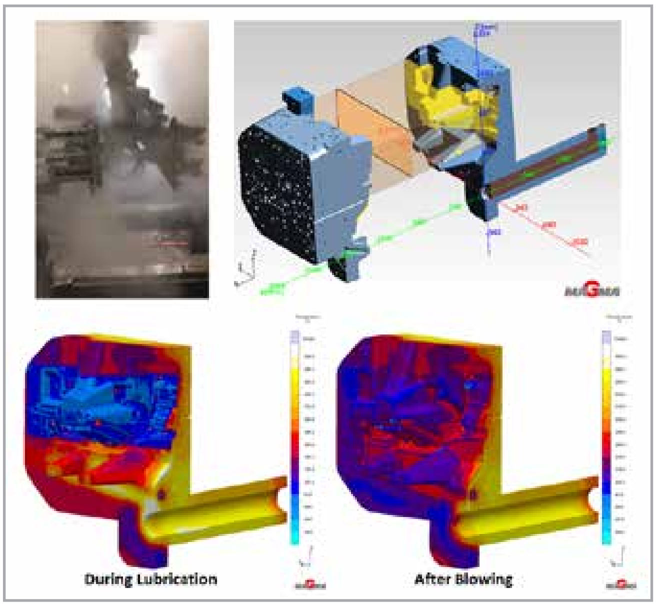


Fig. 2 - Temperature distribution in the die parts during lubrication and after blowing.

The thermal analysis can be combined with an analysis of the tensions and deformations in the part while designing the mold to identify which crucial steps of the production process contribute most to the deformations and tensioning of the casting, as shown in the following images (Fig.3 to Fig 6).

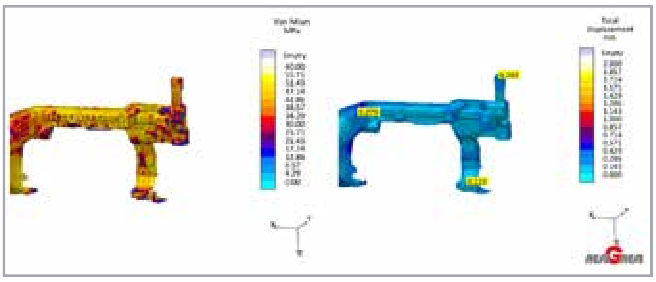


Fig. 3 - Stress distribution and deformations before extraction

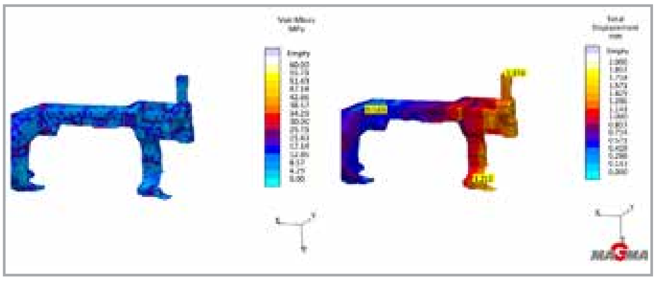


Fig. 4 – Stress distribution and deformations after extraction

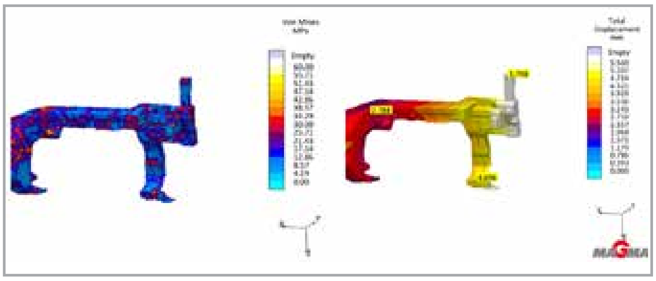


Fig.5 – Stress distribution and deformations after cooling in water

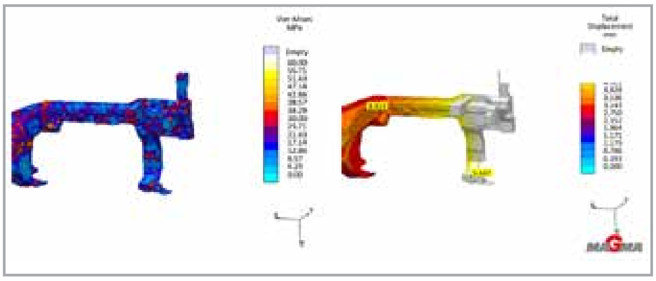


Fig.6 – Stress distribution and deformations after blanking

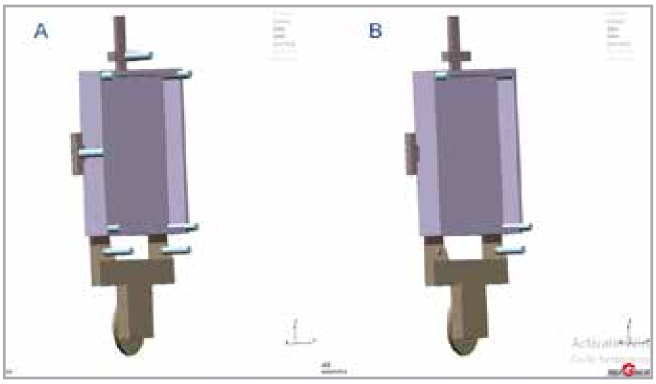


Fig. 9 – Extractor positioning: configuration A and B

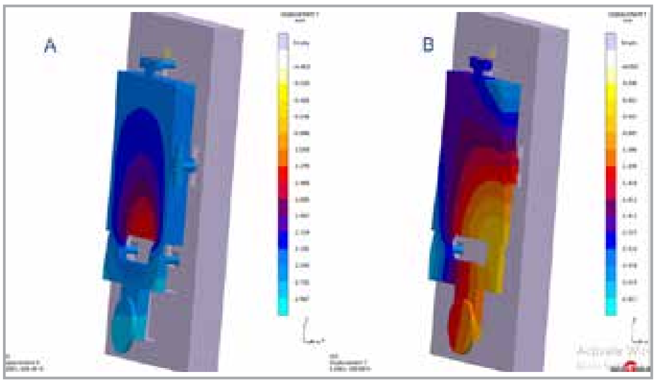


Fig. 10 – Deformed component: configurations A and B of the extractors

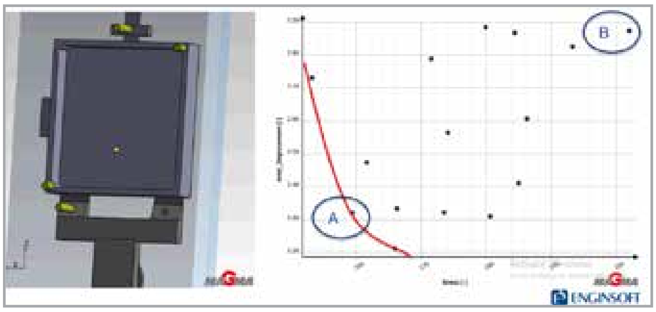


Fig. 8 – Optimizing the positioning of the extractors

Consequently, it is possible to virtually evaluate the effects of modifications to the geometry of the component rather than to the equipment used to produce it. In particular, the casting and venting system can be modified, or the heat of the mold can be adjusted by varying the efficiency of the thermoregulation system and the cycle times that govern the opening, lubrication and blowing phases.

The extraction process can also be evaluated by examining the contact pressure between the part and the mold and the pressure exerted on the component by the extractors (Fig.7), to evaluate the efficiency of the various extractors during the thrust phase and to identify which are key to eject the casting from the molds. In addition, a virtual optimization of the number and position of the extractors can be added to reduce some deformation problems caused by extraction (Fig. 8). For example, if we observe the positions of the extractors in the two extreme configurations A and B, we can see the resulting deformation of the component, which is much more homogeneous in case A, as shown in Fig. 10.

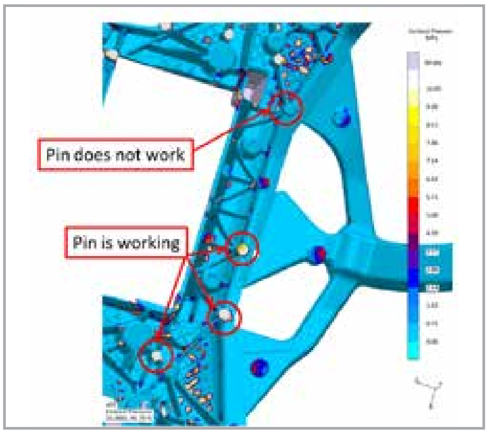
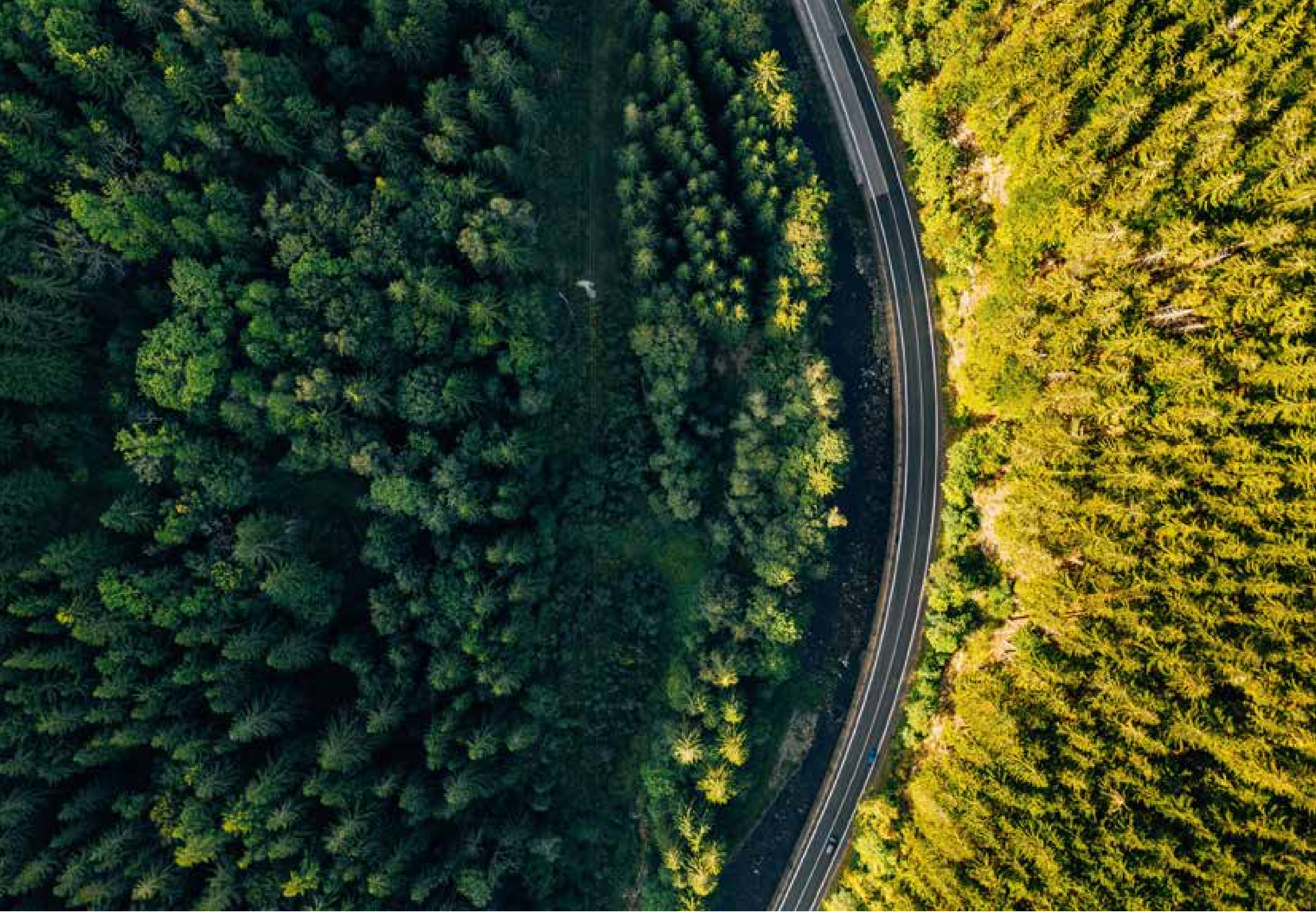


Fig.7 – Contact pressure during extraction

In conclusion, we can affirm that simulation is already becoming an essential tool in the product and process design phases, not only to evaluate the classic defects of the die-casting process (such as air entrapment, shrinkage porosity and cold joints), but also to reduce distortions and residual stresses.

These latter are becoming more and more widespread due to the growing geometrical complexity of the components to be produced for an increasingly competitive market requiring high quality standards, low costs, and reduced time to market.

For more information:
Giampietro Scarpa - EnginSoft
g.scarpa@enginsoft.com



Real-time route optimization of a drone

An Artificial Intelligence application to automatically detect people beneath the drone and to optimally reconfigure its mission

By Giovanna Castellano[1], Ciro Castiello[1], Carmela Agnese De Donno[3], Arturo De Marinis[1], Angelo Emanuele Fiorilla[2], Michele Iacobellis[3], Felice Iavernaro[1], Francesca Mazzia[1], Corrado Mencar[1], Rosa Maria Mininni[1], Andrea Palumbo[3], Gino Perna[4], Gennaro Vessio[1]
1. Università degli Studi di Bari Aldo Moro - 2. Vitrociset S.p.A. - 3. Planetek Italia S.r.l. - 4. EnginSoft S.p.A.

Current regulations place various constraints on the flight of autonomous drones, e.g. prohibiting them from flying over concentrations of people. We report an implementable workflow for drones that automatically detects people and/or crowds in the underlying territory and consequently optimizes the flight plan to minimize the risk of impact with a crowd during an emergency landing.

The greater availability of sensors and devices suitable for drone payloads, which can capture increasingly precise and targeted information and transform it into digital data for processing, has exponentially increased the number of potential applications for drones in both the military and civilian sectors. Among the potential civilian applications are: environmental monitoring, monitoring of areas affected by natural disasters such as earthquakes, floods,

Fig.1 - The route optimization workflow

and fires; search and rescue operations in emergency situations; remote sensing to create maps identifying areas of interest, etc. Drones are mobile devices that can reach relatively high speeds by moving independently through the air, thus offering the advantage of movement that is unrestricted by the characteristics of a given territory or route. To enable automatic piloting, these devices must be equipped with appropriate sensors that acquire and communicate data to special processing units which run specific software to process the data in order to make piloting decisions to achieve a high-level goal (mission).

The drones' missions are subject to restrictions imposed by European Union Aviation Safety Agency (EASA) as well as to environmental and territorial limitations. These are particularly important because of the increasing use of drones in Urban Air Mobility tasks, which require strict safety constraints for their operation. In particular, the European Commission Implementing Regulation (EU) 2019/947 "on the rules and procedures for the operation of unmanned aircraft" prescribes, for the Open Category of drone operations, that "the remote pilot [shall] ensure that the unmanned aircraft is kept at a safe distance from people and that it is not flown over assemblies of people".

Nevertheless, unforeseeable problems, such as sudden adverse weather conditions, GPS failure or degradation, loss of data link with the remote pilot, etc. may occur which could still cause the drone to fly over assemblies of people. The goal of the research is to develop a solution to enable drones to:

1. automatically detect people and/or crowds in the territory beneath them;

and consequently to

2. optimally redefine the flight plan (mission) to avoid the people.

These operations are necessary to adapt the flight path in real time to bring the drone into safe areas so that the risk of impact with people during an emergency landing is minimized. This challenge led to the development of a research project being conducted jointly in partnership between the Italian universities[1] and the top-level industrial partners[2] in the RPASInAir project[3], which is co-funded by the European Structural and Investment Fund (ESIF) and the Italian "Programma Operativo Nazionale (PON) Ricerca e Innovazione 2014-2020" fund. This research project is affiliated to the growing research infrastructure of the "Taranto Grottaglie Airport Test Bed"(IATA: TAR; ICAO: LIBG)[4].

The proposed workflow

Analysis of the problem led to the definition of a workflow for path optimization that includes the following steps:

1. Planning the initial drone route based on the goals and constraints of the mission;

Fig. 2 - The prototype drone is an eight-copter unmanned aerial vehicle (UAV) equipped with a gimballed camera and on-board computing capabilities

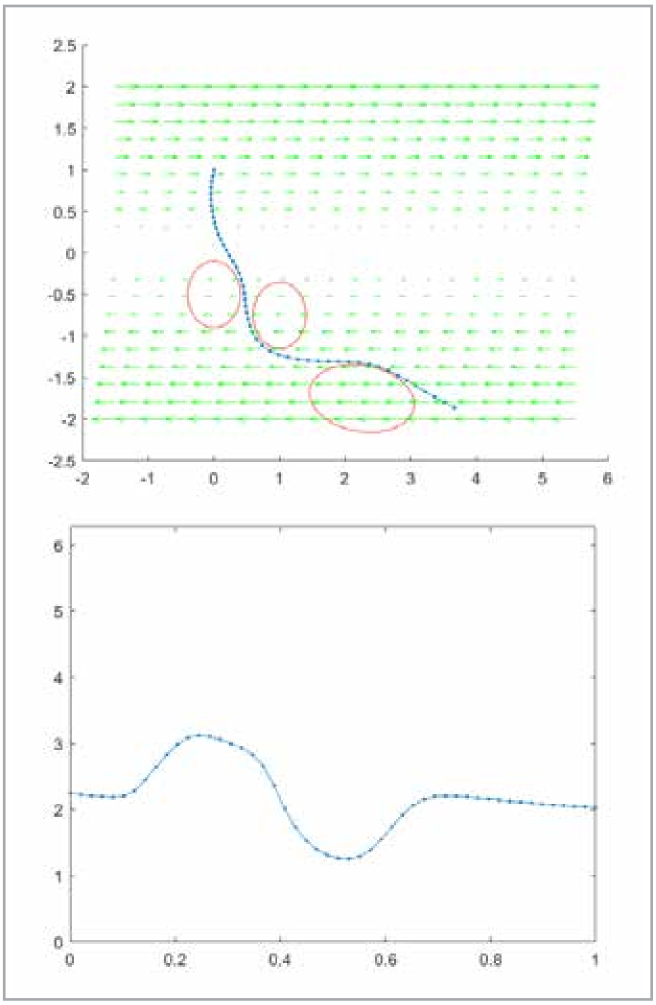


Fig 3 - a) (Left) Optimal trajectory in a windy area with three obstacles . b) (Right) Yaw angle as a function of time to achieve the optimal trajectory .

2. Continuous acquisition of images and telemetry by the drone's sensors;
3. The generation of bounding boxes to detect individuals of terrain to be avoided;
4. The use of geo-referenced images, bounding boxes, and telemetry to provide images and telemetry with spatial and temporal metadata;
5. Re-routing of the flight based on recognized ground patterns to be avoided.

The prototype drone to be used for the final tests is a multi-rotor aircraft equipped with eight engines/propellers, with a load capacity of not less than 1.5 kg, and a variable range depending on the total weight, but in any case not less than 15 minutes. Intermediate tests will take place in a simulated environment.

The workflow is executed by a network of software modules, which are interconnected through efficient inter-process communication technologies. The Manager module allows the on-board high-level algorithms and applications to interface with the ground control station (GCS). The software enables different flight and/or mission functions to be controlled, as well as receiving periodic telemetry messages from the ground control station. The Manager module also receives an initial flight plan as a list of waypoints from the GCS.
This flight plan is the result of an optimization process that takes into account the presence of interdicted areas, fixed obstacles, etc. It is realized using mathematical methods that follow the "optimize then discretize" paradigm.

These are based on the variational calculus and transform the problem of optimal control into a Hamiltonian boundary values problem (BVP). More specifically, the cost function in the minimization procedure is the time required for the drone to traverse the given waypoints while avoiding a number of forbidden zones, assuming a constant cruising speed. The module converts the geodetic coordinates of the waypoints into their universal transverse mercator (UTM) or planar form to make them compatible with the BVP code; it also manages the presence of wind as an input vector field. The image on the left in the figure below shows an example of an optimal flight path (solid line) joining two waypoints while avoiding three areas (encircled by ellipses) that are under the influence of a wind that has a constant direction but variable intensity (green arrows). The picture on the right shows the course angle as a function of time which corresponds to the optimal trajectory, and which is also available as an additional output of the module.

The resulting waypoints are expressed as geodetic coordinates. Trajectories based on algorithms such as minimum jerk splines and Hermite cubic splines are generated from the list of waypoints and the specific time in which each waypoint must be reached. The choice of the trajectory model depends on specific objectives, such as reducing battery consumption and oscillation phenomena, or minimizing the deceleration and acceleration at waypoints by using flight profiles that are very close to those of fixed-wing drones.

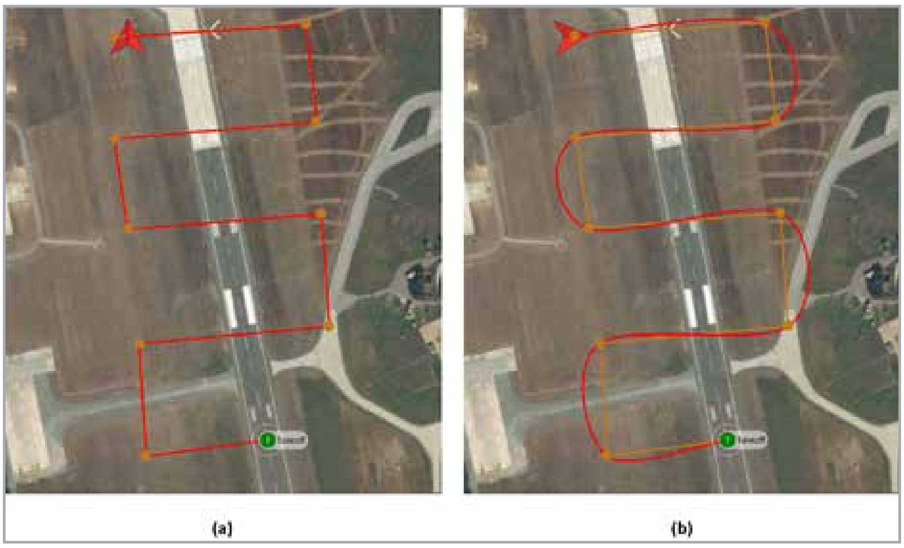


Fig. 4 - Two different trajectory models can be chosen: (a) minimum jerk trajectory, to reduce battery consumption and oscillation and (b) Hermite cubic splines to mimic the flight of a fixed-wing drone


Fig. 5 - Example of pedestrian detection using YOLO on an image frame from a benchmark dataset

The Manager module also collects information from the camera and other sensors for the subsequent stages of the workflow. A specific module synchronizes the images captured by the camera with the position (image geotagging) and the drone's heading information relative to the time stamp of the snapshot. The camera is mounted on a stabilized 3-axis gimbal to attenuate vibration noise during flight and is constantly oriented in a nadiral position.

The module for pedestrian detection receives the images from the Manager module and returns a set of bounding boxes concerning the risky zones to be avoided during flight or landing. Since the input consists of images in the three RGB channels taken from the camera, the most effective and efficient approach to use for detection is based on Convolutional Neural Network models, a particular class of feed-forward artificial neural networks that are at the forefront of many Computer Vision tasks, including object classification and recognition; facial recognition; pattern recognition in video; etc.

Object detection is the recognition of a variable number of objects within digital or video images, along with an accurate estimate of their location within the image. This localization is achieved by predicting the coordinates of the bounding boxes that surround each detected object. To satisfy the need for real-time responses in a limited computing environment (such as that of a drone) without sacrificing detection accuracy, the TinyYOLO model is used . "You only look once" (YOLO) is a simple approach based on a single convolutional network that provides excellent predictive abilities and detection speeds with real-time object detection.

The bounding boxes returned by the detection module are geo-referenced in order to obtain geodetic coordinates. This step is performed by applying the typical logic of a 3D rendering pipeline, like the one found in near real-time simulation (for example in 3D games), appropriately inverted to transform the image coordinates into world coordinates.

The bounding boxes serve as input to the real-time optimization module, which generates a new flight plan by modifying the original one to avoid the areas represented by the bounding boxes. The selection of the optimal path is inspired by algorithms drawn from robotics and interactive games.

Basically, the problem of optimization requires the system to find the best route from one point to another (in our case from the drone's current position to the next way-point) keeping in mind the forbidden zones and the travel time. The approach follows graph theory and is based mainly on three algorithms: (i) Breadth First Search, which is important for finding a route, and for generating procedural maps, distance maps, flow paths and other types of map analysis; (ii) Uniform Cost Search, which identifies the priorities on the routes to be analyzed; (iii) A* search algorithm, which is an informed search procedure that is able to optimize the cost of a route by performing an efficient search within a wide range of solutions.

Preliminary experiments show that the workflow for route optimization is promising both in terms of effectiveness and efficiency. The most time-consuming modules, in particular the detector module, are able to perform accurate detection in about 100ms and the subsequent A* algorithm takes some extra 30ms (reference hardware platform ODROID – XU4).
However, the research project is an ongoing activity that requires several stages of development and testing, which are the main focus of the ongoing research efforts.

Acknowledgments
This work was supported by the Italian Ministry of Education, University and Research within the RPASInAir project with Grant PON ARS01_00820.

References
[1] University of Bari Aldo Moro and Polytechnic of Bari
[2] EnginSoft S.p.A. (www.enginsoft.com), Vitrociset S.p.A. (www.vitrociset.it), Planetek Italia S.r.l. (www.planetek.it), Leonardo S.p.A. (www.leonardocompany.com)
[3] RPASinAir – Integration of remotely piloted aircrafts in non-segregated airspace
[4] See also A. Zilli & R. Guida, "The Grottaglie Airport Test Bed: The need for an Experimentation Facility", ES Newsletter 17(1), Spring 2020 https://www.enginsoft.com/assets/pdf/newsletter/newsletter2020_1.pdf

For more information:
Corrado Mencar - Università degli Studi di Bari A. Moro
corrado.mencar@uniba.it

Digital platform for circular economy in cross-sector sustainable value networks

By Giovanni Borzi, Anteneh Yemane, Chiara Salatin
EnginSoft

The circular economy and the role of research and innovation

In a world projected to have close to nine billion people by 2030, the challenges of expanding the supply of resources to meet future demand are unprecedented. Unless we rethink the way society uses materials in the linear economy, resources that are vital to industry could run out within the next five to 50 years, and reach planetary boundaries with irreversible consequences. A new industrial model that decouples revenues from material input, and production from resource usage, is essential to achieving sustainable development, both in the early-industrialized countries and in emerging economies.

The "Circular Economy" (CE) is an emerging paradigm that aims to establish a new roadmap for sustainable development by decoupling economic growth from resource consumption. Potentially, this could generate new business opportunities in worldwide economies and significantly increase resource efficiency in industrial systems. The vision for the circular economy is to radically change the current linear "take-make-dispose" economic approach, which generates massive amounts of waste. Instead, the CE model adopts restorative and regenerative approaches, both by intention and by design.

Currently, CE is implemented by collecting post-use products from the market through reverse logistics systems and conducting

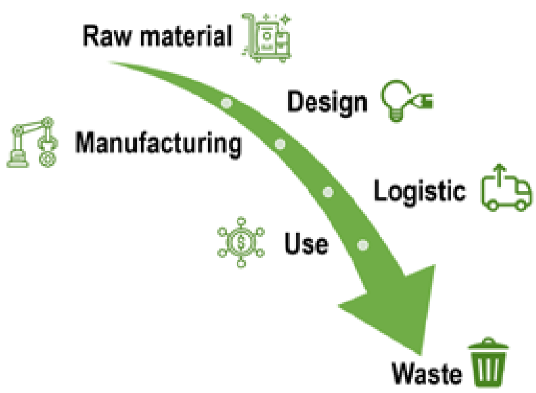


Fig. 1 - From linear economy…

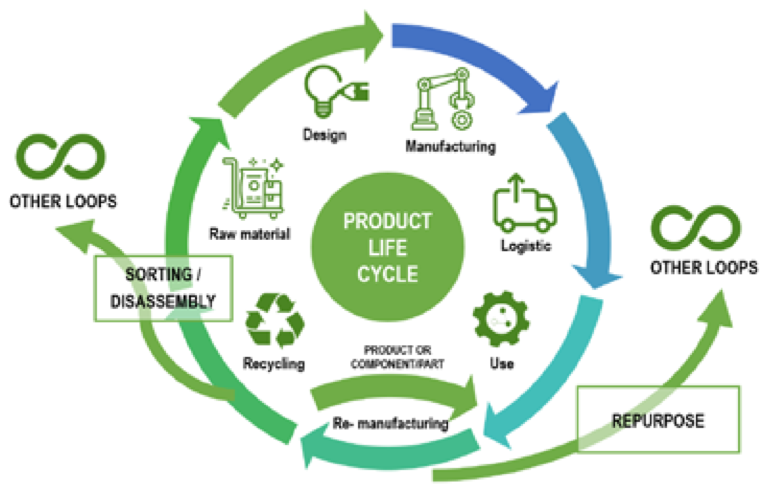


Fig. 2 - …to circular economy

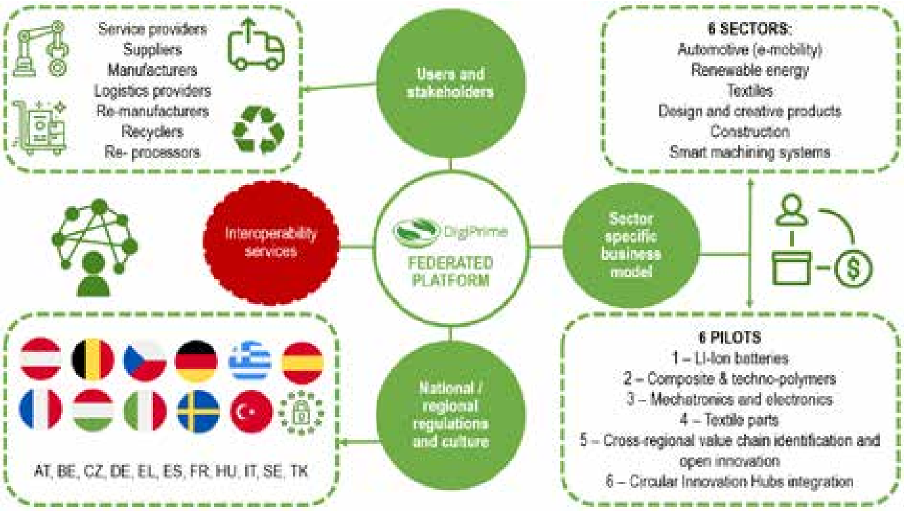


Fig. 3 – Map of the DigiPrime action plan

demanufacturing-remanufacturing operations that are only sustainable for simple products and for specific materials. These operations are carried out individually in "silos", without the sharing of information and economic benefits: this can only be overcome by rethinking the current European industrial system in terms of new collaborative and sustainable value networks.

DigiPrime, funded by the European research and innovation programme, Horizon 2020, and led by the Politecnico of Milan, aims to develop a new concept for a circular economy digital platform to provide timely and user-specific information among the value-chain stakeholders of different sectors and production loops. It will thus support new circular business models based on the data-driven recovery and re-use of functions and materials from high value-added products and components.

DigiPrime will create and operate a federated model of digital platforms for cross-sector business in the circular economy. The nodes in the federation will offer interoperable functions and data, accessible to other nodes, and combined with local data and services; connectors and open interfaces will enable easy integration of new services from third parties. Particular attention will be paid to the creation of reliable data sharing mechanisms, thereby preserving the confidentiality of business-critical data.

The federated platform promotes the creation of cross-sector circular value chains where the residual value of a product post-use is maximized by establishing alliances among users and stakeholders from six different sectors, and by transferring the product components and materials into applications with differing requirements and national regulations. Four pilots in specific innovative circular business cases connecting six key sectors, namely automotive (e-mobility), renewable energy, textiles, design and creative products, construction and smart machining systems will demonstrate the platform's benefits. One pilot project will be dedicated to the consolidation of a new systematic approach to identify cross-regional, cross-sector value chains in Europe, while another will ensure the DigiPrime platform's effectiveness in supporting the integration of the European networks of the Circular Innovation Hubs.

EnginSoft's role in the project

EnginSoft is contributing to the formalization of the digital platform's services and to the development of specific technologies applicable to individual services.

At the digital platform level, EnginSoft has formalized all the services to be embedded in the DigiPrime platform, covering both services at the value-chain level and at the operational level. These formalizations form the basis for the detailed technical specifications and future developments. During this process, EnginSoft gathered a large amount of cross-sector knowledge drawn from several experts to construct a coherent framework of services to be offered through the DigiPrime platform. A co-design approach was adopted involving circular economy experts with regard to:

- strategically important products such as batteries, and electronic components containing rare earth materials,
- suppliers of digital platform technology,
- business experts, and
- involving the "owners" of the industrial pilot projects in different forums to ask their input and to verify the final designs of the service.

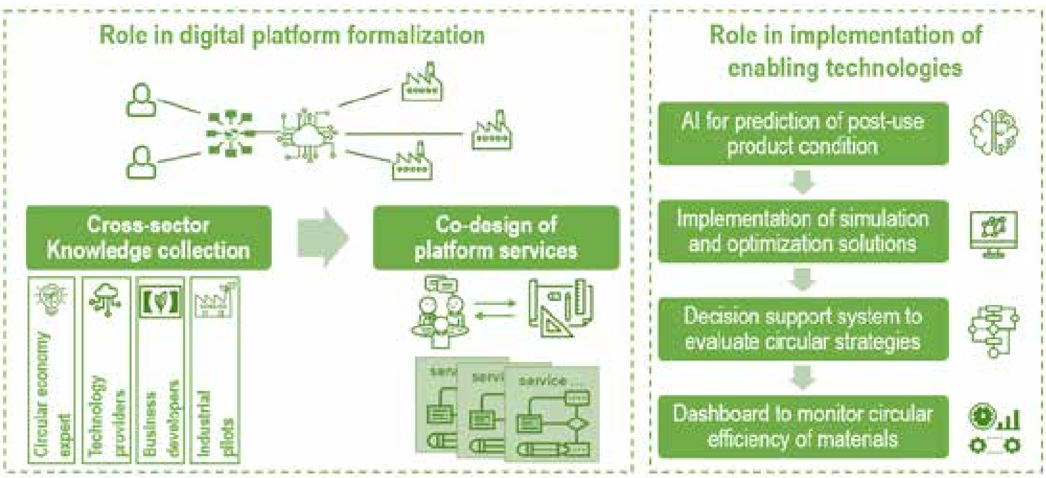


Fig. 4 – The main areas of EnginSoft's contribution to DigiPrime

EnginSoft is also involved in the development of digital solutions for individual services. Significant developments include:

- AI-based applications to predict the condition of post-use products before they enter the de- and remanufacturing processes.
- Implementation of simulation and optimization solutions to support performance analysis and optimization of the reverse-logistics configurations by maximizing the economic and environmental attainments.
- Decision support systems to analyze the circular strategy and define the best actions for specific post-use product conditions in the context of the circular economy.
- Dashboard to monitor the circular efficiency of materials, which will be used to group the system's key performance indicators (KPIs) and their visualization.

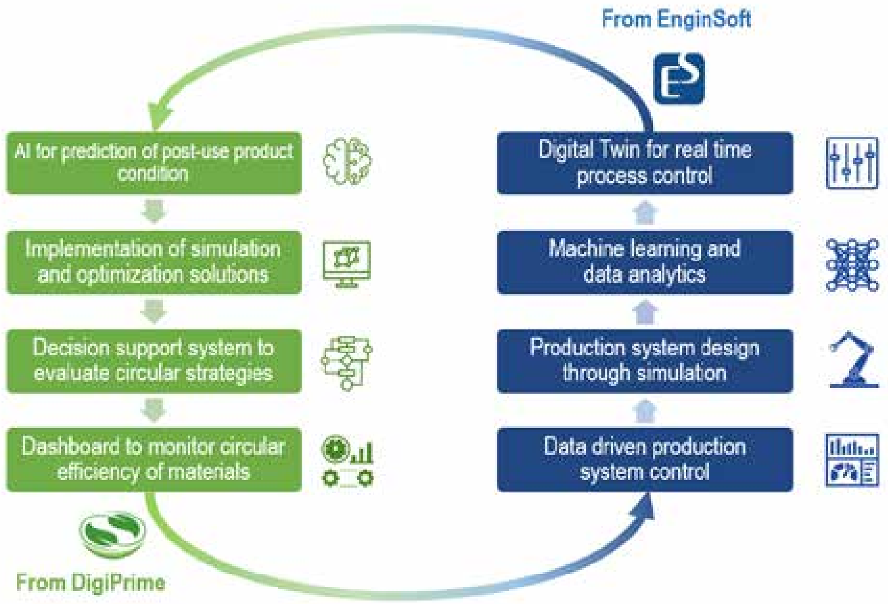


Fig. 5 – Synergetic loop between EnginSoft's expertise and DigiPrime's innovations

In addition to these main areas of development, EnginSoft is providing expertise in various technical activities related to digital manufacturing technologies as part of the circular economy.

Project synergy with EnginSoft's activities

Digital manufacturing and the use of the Digital Twin have been identified as key areas for potential synergy with DigiPrime innovations. The goal is to crosspollinate the best practices in digital manufacturing technologies and transfer them to address the challenges in enabling sustainable circular economy strategies. Some of the synergetic links identified are model-based real-time process control, machine learning and data analytics to support the AI for predicting post-use product condition.

The techniques and simulation tools used in production system design and control have promising potential for the simulation and optimization needs of circular reverse-logistics solutions. Specific tools from the EnginSoft product catalogue can be integrated into the DigiPrime platform test bench during the validation and verification phases.
On the other hand, DigiPrime innovations have the potential to extend existing solutions in the engineering and control of production systems, in particular to address the challenges of industries working in the circular economy.

The next generation of simulation and optimization solutions will aim to address the reverse logistics of post-use products, which are core features of the circular industries.

Activities and progress

DigiPrime started in January 2020 and, despite the interruption caused by the effects of COVID-19, the partnership has managed to begin implementing the project.

A detailed formalization of DigiPrime's digital services has been conducted, taking into account the specific characteristics of each pilot project and the needs of the different users, which provides an important roadmap to guide the development, implementation and effective exploitation of the DigiPrime service eco-system. EnginSoft, together with the partners, has collected and processed information about:

1) the technical needs of each pilot project,
2) future scenarios and changes in circular business processes (both at operational and value-chain level) and
3) the roles and expectations of users when using DigiPrime digital services.

The six DigiPrime pilots were classified into two main groups – technical and non-technical pilots – based on their characteristics. The first group is oriented towards technical processes and stakeholders (Pilot 1 to Pilot 4), while the second category (Pilot 5 and Pilot 6) is oriented towards the organizational and innovative aspects of circular business management. The main stakeholders identified are the actors and the potential beneficiaries of the DigiPrime services, and their impact on circular business was explored to improve the architecture design and the integration of the digital platform.

Future prospects and expectations

The European Green Deal places the concept of the circular economy at the center of efforts to transform the European Union into a fair and prosperous society where economic growth is decoupled from the use of resources.

About DigiPrime

Duration: 2020 -2023

Partners:
POLITECNICO DI MILANO, ATLANTIS ENGINEERING AE, BALANCE TECHNOLOGY CONSULTING GMBH, BOSCH REXROTH AG, CENTRO TESSILE COTONIERO E ABBIGLIAMENTO SPA, CIRCULAR ECONOMY SOLUTIONS GMBH, COBAT SERVIZI, CONTINENTAL AG, DESIGNAUSTRIA (DA), E REPAIR SRL, EDAG ENGINEERING GMBH, ENGINSOFT SPA, ENGINSOFT TURKEY MUHENDISLIK YAZILIM TICARET LIMITED SIRKETI, ENVIROBAT ESPANA SL, EXTRA RED SRL, FLEXIS AG, FUNDACIÓN TECNALIA RESEARCH & INNOVATION, HOLONIX SRL– SPIN-OFF OF POLITECNICO DI MILANO, IDEA STRATEGISCHE ECONOMISCHE CONSULTING, INDRA SAS, INGENIERIA Y APLICACIONES SOLARES 2005 SL, INNOVA SRL, INOTEX SPOL SRO, KARLSRUHER INSTITUT FUER TECHNOLOGIE, KNORR-BREMSE SYSTEME FÜR NUTZFAHRZEUGE GMBH, LULEA TEKNISKA UNIVERSITET, MAGYAR TUDOMANYOS AKADEMIA SZAMITASTECHNIKAI ES AUTOMATIZALASI KUTATOINTEZET, NTUA – NATIONAL TECHNICAL UNIVERSITY OF ATHENS, PLASTIPOLIS, RESEARCH AND EDUCATION LABORATORY IN INFORMATION TECHNOLOGIES, RIVIERASCA SPA, SAUBERMACHER DIENSTLEISTUNGS AG, SIEMENS GAMESA RENEWABLE ENERGY INNOVATION & TECHNOLOGY S.L., SIMPLAN AG, TTS – TECHNOLOGY TRANSFER SYSTEMS SRL, UNIVERSITAT AUTONOMA DE BARCELONA, VELTHA IVZW.

DigiPrime

Website: www.digiprime.eu

"The project leading to this article has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 873111".

According to the ambitious Circular Economy Action Plan adopted in March 2020 by the European Commission, realizing the circular economy necessitates more than just traditional R&D.

It will involve changes in entire systems as well as combined effort by researchers, technology centers, industry and SMEs, the primary sector, entrepreneurs, users, governments and civil society. It also needs enabling regulatory frameworks and further public and private investment.

Recognizing the key role of research and innovation, the European Commission has adopted an action plan to stimulate Europe's transition to this new model. It covers the whole cycle: production, consumption, waste management and secondary raw materials.

There are further links to EU policy on raw materials and forthcoming circular economy initiatives on plastics, water and the interface between waste, products and chemical policies.
Critical innovation in services, in business models, and in integrating digital technologies is also necessary to support the deep economic and societal transformation required.

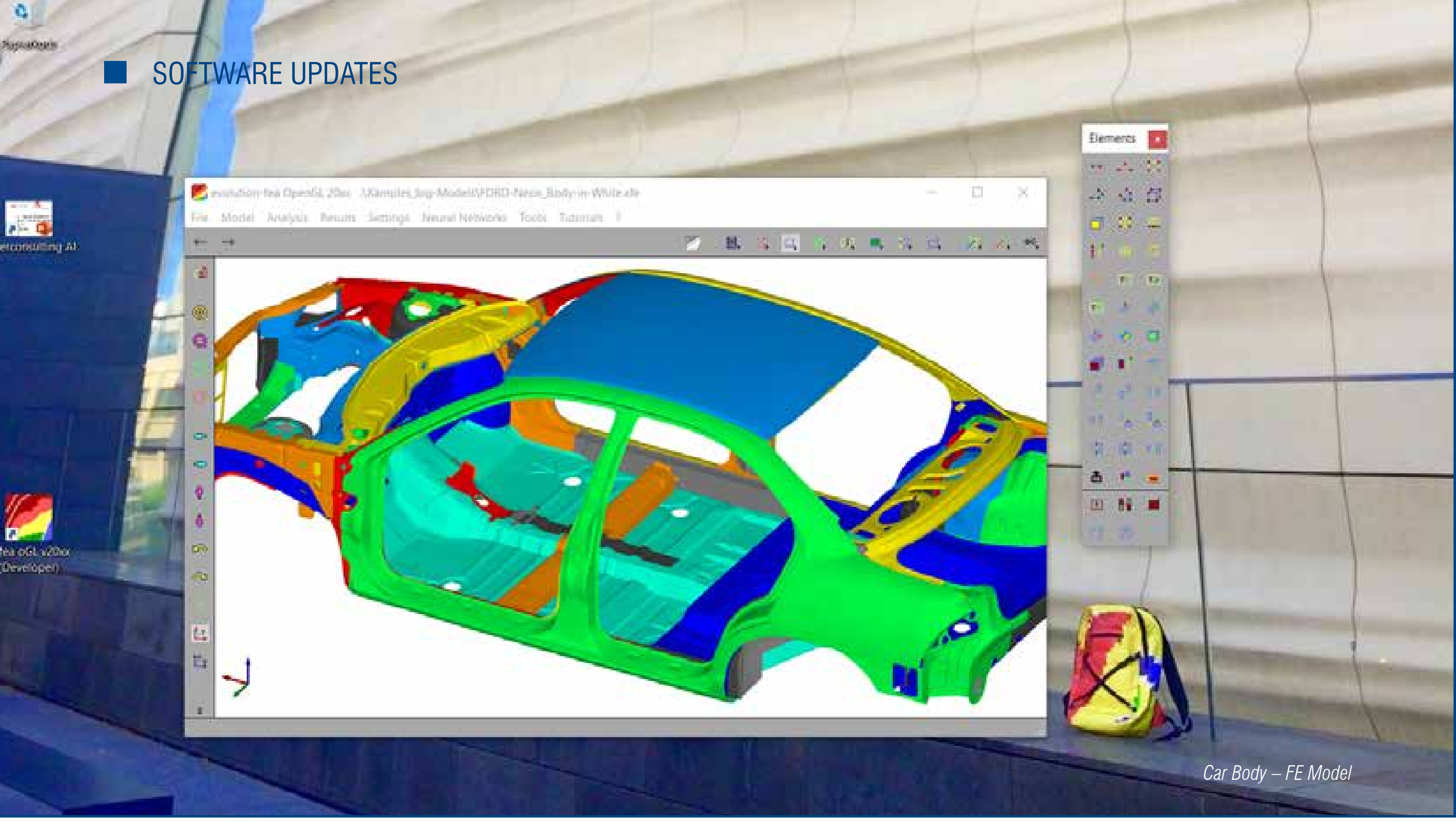
During the project, the DigiPrime platform will be implemented and tested in the six pilots, to demonstrate the effectiveness of the individual services and the value of cross-node-enabled cooperation. Open Calls will enrich the service catalogues and extend their validation. DigiPrime partners have a strategic interest in improving the platform to create a market-ready offer.

Thanks to the SEED programme, a global partnership to promote entrepreneurship for sustainable development, the platform will be offered to organizations interested in creating new nodes in the federation. Platform tool and service developers plan to involve technical organizations that are interested in creating high added-value tools and applications to be added to the platform (i.e. starting from the core platform services).

The targeted services and experiences, developed by Digiprime, are therefore in line with the key findings of the Workshop "Innovative services and products for the circular economy" held in March 2020 by the Executive Agency for Small and Medium-sized Enterprises (EASME):

- A substantial impact will only occur if a systemic approach is taken together with the active participation of stakeholders along the entire value chain, including the full involvement of industry.
- A paradigm shift is needed to widely integrate "circular thinking" to support the development of innovative products and services. Eco-design for recycling, reuse and repair is critical to achieving sustainable and circular products, but it is not enough, and attention needs to be paid to developing solutions for end-of-life products. Certification and standardization activities are key to promote sustainable product design and value chain management practices and to bring solutions closer to the market.
- Digital tools, such as platforms, big data, and block chain technology among others, are enablers for business model innovations to ensure data traceability and transparency, as well as to engage stakeholders throughout the value chain.

For more information:
Giovanni Borzi - EnginSoft
g.borzi@enginsoft.com



Car Body – FE Model

Open source simulation: the key to quality and freedom

Choosing open source offers lower TCO and greater ROI

By Erich Payer
payerconsulting GmbH

This article for business readers makes the case for choosing open source software as opposed to proprietary software due to the many benefits it ensures to users, including better quality, more flexibility, reduced costs, and an end to vendor lock-in.

What is simulation?

"Simulation is an act of predicting the future."

Simulation is similar to prototype testing, but it is done with computer software in far less time and at much lower cost – well before the product is manufactured. Simulation reduces or even replaces the costly and time-consuming process of producing and testing physical prototypes. It allows product features to be systematically modified to improve critical product attributes soon in the design-build cycle, enabling tooling and material decisions to be made early. Simulation supports designers' understanding of the design tradeoffs and in making the necessary decisions to optimize their products. Companies are therefore able to design and build products that are lighter, easier to manufacture, sell, service and support, get to market sooner, perform better, and have lower warranty costs.

"Simulation leads to decreased product development costs, increased product sales, and increased profits!"

Toyota study

The main benefits of using simulation in product development have been documented in a study recently published by the Japanese car maker Toyota.

This study documents that for Toyota's "Avensis" passenger car model, overall development time was able to be reduced by 20%. Development costs were decreased by 30%, and 65% fewer prototypes had to be built for physical testing prior to series production.

Open source kernel 'e-fea'

The Open Source Finite-Element software evolution-fea (e-fea) has been developed by engineers for engineers to combine

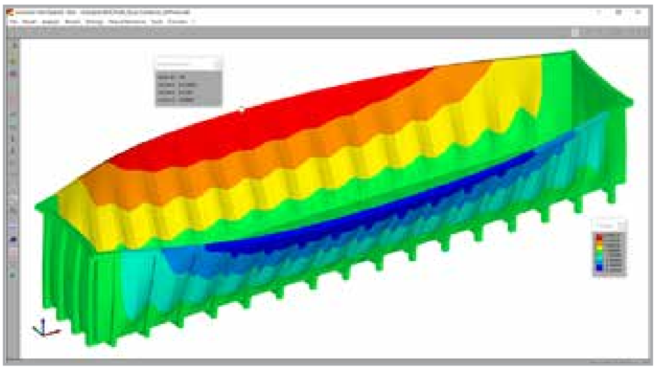


Fig. 1 – Stiffness of a chemical fluid container

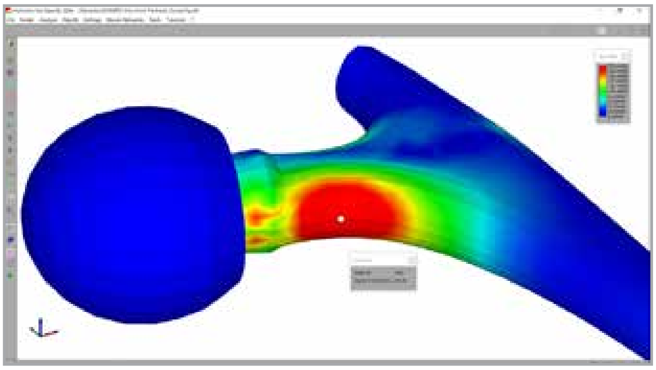


Fig. 2 – Durability of a hip joint prosthesis

technology with usability and is now being successfully used by numerous market leaders in various industries worldwide (see figs. 1-5).

The closed source vs. open source model

All software applications are built from source code, namely the numerous lines of instructions that programmers write for computers to interpret and execute.

A closed source or proprietary model, which is the traditional model of software distribution, is one where the source code remains private information. Proprietary software is distributed as object or binary code that reveals its inner structure only through reverse engineering based on a difficult analysis of CPU instruction sequences. Users or third-party developers who wish to modify the software are prevented from doing so by intellectual property laws, even when their reverse engineering efforts are otherwise successful.

Open source distribution models invert this practice. The source code is not secret, but is made freely available for anyone to download, redistribute, view, and alter at their own discretion. If you can program in the language in which a particular open source application has been written, and have enough time available, you can edit the source code to make the software behave exactly as you wish. Furthermore, you are under no obligation to share your enhancements with anyone.

The freedom to run the software as desired, to study and modify the source code, to make and redistribute copies, and to publish modified versions is absolutely crucial. This means that users and

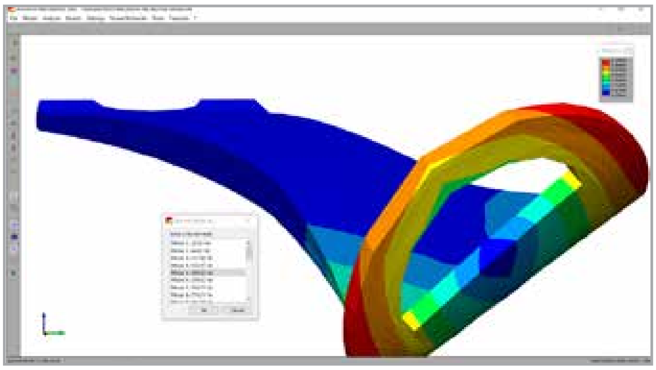


Fig. 3 – Natural vibrations of an Alpine ski

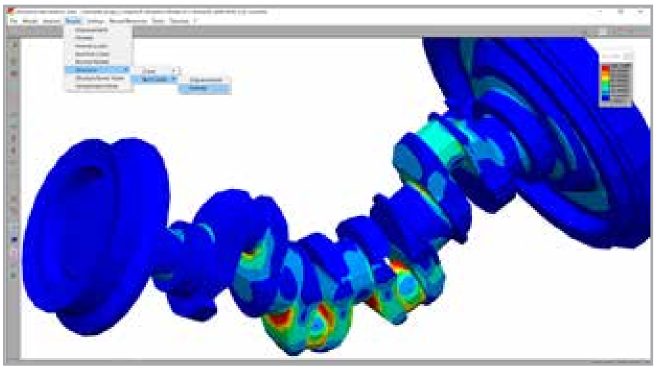


Fig. 4 – Nonlinear dynamics of an I4 crankshaft

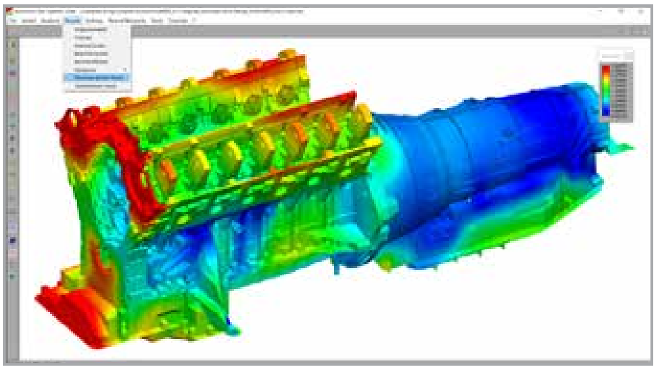


Fig. 5 – Structure born noise at a V12 engine

developers are the stakeholders in open source software.

"Open Source is about sharing ideas, code and innovation!"

The benefits of open source

"Open source creates a community to which all constituencies can contribute and from which everyone can benefit."

In the open source model, the source code of the software is made freely available. These models therefore leverage the collective wisdom, experience and expertise of developers to rapidly meet the requirements of the most demanding software users. Its exponents gather in a global open source community and are keen to work together to support each other and improve the software, which is used for personal or business purposes.

They collaborate to develop software improvements, offer customizations, provide benchmarks, and provide support and services.

"The result is that open source models promise better software quality, higher reliability, greater flexibility, lower costs, and an end to predatory vendor lock-in!"

Costs

Under the given economic boundary conditions, enterprises seeking to cut costs are increasingly drawn to open source software, according to Gartner (see Fig. 6).

Open source software can essentially be obtained free of charge or at very low cost. Similar to proprietary software, there can be costs for training, consulting, maintenance etc. However, the decisive factor is the total cost of ownership (TCO), where there is usually a significant price difference between a closed source and an open source solution.

All Amounts in €	Year 1	Year 2	Year 3	Total
Investment	105.000 25.000	15.000 5.000	15.000 5.000	135.000 35.000
Savings				
- Prototypes	35.000 75.000	35.000 75.000	35.000 75.000	105.000 225.000
- Measurement/Testing	10.000 25.000	10.000 25.000	10.000 25.000	30.000 75.000
- total Cost Reduction	45.000 100.000	45.000 100.000	45.000 100.000	135.000 300.000
Contribution to Profit	(60.000) 75.000	30.000 95.000	30.000 95.000	0 265.000
Payback Period				3 Years 3 Months

} lower Investment Costs by Open Source

} higher Savings due to enhanced Software Features

} Faster Payback & increased Contribution to Profits

Fig. 6 – Return on investment (ROI) comparison for proprietary vs. open-source FEA software

Among the arguments for the low TCO of open source software are:

- potentially zero purchase price
- potentially no need to account for the number of copies in use
- high competition in the market, reduces vendor lock-in and monopoly pricing
- longer uptimes with reduced need for expensive system administration
- protection against the need to adapt an internal IT strategy to the cash needs of a software vendor

Open source software is developed and maintained through community forums. Developers volunteer their time and expertise and are coordinated by smaller number of paid programmers.
The lower overhead costs translate into substantial savings in development costs. According to Accenture Australia, using open source software as the basis of your development can reduce your development time and budget by 50%!

Licensing

First of all, there are no licensing fees! And open source means that you have the source code –now and forever!

You can place your open source software wherever you want it, whenever you want it. You can embed it in your products, or ship it with them to help your customers and to improve the value of your product without sacrificing your price advantage.

Freedom from proprietary lock-in

Every business should try to avoid being tied to a specific supplier or becoming overly dependent.

The strategy that makes it difficult, or impossible, for software users to switch suppliers is known as "proprietary lock-in". This strategy has been successfully pursued by commercial software vendors, giving them the power to increase the price of product upgrades or support without excessively risking the loss of existing customers.

Commercial software vendors use a number of tactics to persuade their customers to upgrade, more or less willingly. Typical tactics include moving to allegedly "new and improved" file formats (which require the "new and improved" software to read them), or withdrawing support and bug fixes for older versions after a short period.

Choosing to use open source software counteracts the pressure to upgrade for a vendor's commercial ambitions.

Mitigation of supplier failure or product discontinuation

Commercial software vendors go bankrupt or get bought out from time to time, or they may arbitrarily decide to cease development of a product.

In this case there is no guarantee that their software product will continue to be available, supported, or updated. Even with solid companies, new releases often result in the discontinuation of, or ceasing of support for, older software versions.

This can force users to change products, which can be very expensive and difficult, especially if they are heavily locked-in to their current product.

With open source software, this danger is greatly reduced. The source code is not "owned" in the same way that proprietary source code is. So, if a vendor goes out of business, the source code can be taken and developed by anyone with an interest in the software's survival.

Choice

Open source supports choice. The open source software's source code may be operated and maintained by multiple vendors.

Thus, a user can easily choose among different providers for improvements, customization services, training, and support.

Testing and evaluation

Open source software can be extensively test driven before procurement without the need for problematic evaluation or time limitations. There are no rules on restricted use that limit the number of evaluators. Anyone who is interested can get involved. And, there is no messy legal paperwork to process, or permission to seek.

With access to the source code, users of a wide variety of platforms, operating systems, and compiler combinations can compile, link, and execute code on their system to test portability.

Benchmarking

Benchmarking is an important activity in evaluating whether a product is suitable for you.

However, when it comes to benchmarking, many proprietary vendors are very selective. They usually present their products in the best light. You also have to sign agreements that prevent you from publishing benchmarks for their products.

"Open-source benchmarks play on the market of free ideas!"

Users and developers of open source software share their benchmarks with the open source community and get help from their peers to build the right kind of benchmark. Porting someone else's benchmark to a specific platform and comparing results helps the entire community to understand possible implementation differences between platforms.

Support

A key advantage of open-source software is that it is always possible to find and retain an individual or an organization from the open source community to provide technical support. Furthermore, since the source code is freely available, you are not limited to obtaining support from the original authors of the code.

Open source community

By adopting open source software, you become part of a global community of users and developers who have an interest in working together to support each other and improve the software.

According to study by Evans Data Corp., more than two million developers worldwide spend at least part of their time working on open source development projects. So there are many very bright developers out there who are willing to invest their time. They are motivated by pride and peer recognition rather than by business plans provided by management or marketing departments.

Most developers want to use the software themselves and prefer robustness before adding functionality. They will probably consider it a "win" if they can reduce complexity and improve the maintainability of the software. Therefore, clean design, reliability and maintainability are highly valued factors. These qualities are rarely high priorities in the product plan for commercial software.

Moreover, when several authors work in parallel, the best-of-crop solution can be selected instead of the only solution, as is more typical for commercial products.

"The open source community attracts very bright, very motivated developers, who are often unpaid but are usually very disciplined. The quality of the software that is produced by the open source community mostly surpasses that produced by purely commercial organizations."

Reliability

Reliability means the absence of defects that cause faulty operation, data loss or sudden failures – what many people usually refer to as "bugs". Almost all software releases contain such bugs.

When a bug is identified in proprietary software, the only people who can fix it are the original developers because only they have access to the source code. Typically, a bug report must be filed, and then there is a delay before the vendor determines if and when to release an updated version.

Open source software is different. Since a large number of users can access and modify the code, bugs tend to be more visible and are fixed more quickly.

"Given enough eyeballs, all bugs are shallow!"

Quite often, mature open source products set new industry standards for "bullet-proofness".

Improvements

With open source, you are not at the mercy of the commercial ambitions of a traditional software vendor for improvements. You are also not limited to what a single supplier believes you need.

Proprietary software vendors must cater to many different companies' demands and, predominantly, their own.

With open source code, you do not have to wait for the vendor to add the features you need. If it is urgent, you can do it yourself. If you add features and submit the software back to the community, many people you don't know - and don't have to pay - but who are very bright, will usually help you to improve it. While they cannot turn bad ideas into good ones, they can turn good ideas into great ones.

On the other hand, you don't have to upgrade if you don't want to, and you do not have to pay for improvements that you do not necessarily want.

In addition, competition is fierce in the open source community, and vendors must compete like crazy to innovate on top of a common base.

Customization

Open source software can be tailored for the way you do business. It can be readily adapted to meet your specific user needs and may be customized by anyone with the requisite skill.

It is usually within the resources of all but the smallest companies to modify open source software to suit their own needs. And, if in-house development skills do not exist, a short email to the open source project's mailing list will generally help you to find a skilled software developer or suitable consultant.

For businesses or educational institutions, the ability to customize source code always provides a great competitive advantage. They can do so themselves rather than having to persuade a commercial vendor to do so on their behalf.

For example, just think of the language of the software's graphical user interface.
Commercial providers of closed-source software are usually not willing to translate their products into less common languages because the market for them would be too small to guarantee profit. With access to the source code, you can easily translate the user interface yourself.

Auditability

Closed-source software forces users to trust vendor claims regarding qualities such as security, freedom from backdoors, adherence to standards, and flexibility in the face of future changes. If the source code is not available, however, these claims simply remain claims.

By publishing the source code, the authors enable users of the software to check that there is a basis for these claims. Whether this takes the form of a cursory and informal inspection or more rigorous auditing, it is clear that without access to the source, third-party inspection is impossible.

Security

Open source means "what you see is what you get". You can inspect the code line by line to ensure that no disgruntled programmer has buried logic bombs, trapdoors, Trojan horses, viruses, or any other nasty surprises in the code.

Access to the source code makes it easier to detect security flaws - and, as mentioned, flaws in open source software tend to be fixed more rapidly.

Configuration management

Open source products often support multiple compilers from which you can choose.

And, if for some reason you do not want to upgrade a platform operating system you don't have to do so in step with everyone else. If you have a specific platform on which you wish the software to run, you can port it yourself – you don't have to follow the crowd.
Alternatively, if you are a platform vendor and you find that your platform is no longer supported, or you think it is poorly supported, you can support it yourself.

Debugging

With the source code available, you can use a debugger to navigate while your own code interacts with that of the open source software. This allows you to isolate bugs, both in your code and in the open source code, more quickly. Going through the open source code with the debugger helps you better understand how the software works.

Third-party tools

Having access to the source code allows developers to provide additional tools and add-on products that improve the code functionality. For example, they can add new solvers, interfaces, meshing techniques and advanced pre- and post-processing functionality.

Revision management and product evolution

Everyone can beta test upcoming releases of open source software. It is not restricted to a privileged few. Furthermore, new versions are released when the community feels the software is ready, not just to meet management's revenue targets.

Learning from others

If you are interested in programming, open source code provides an excellent resource from which to learn, and open source projects provide a practical environment in which to test your skills.

Conclusion

Today's economic, environmental and technological challenges force companies to take appropriate measures to achieve both higher productivity and better product quality. The evolution Open Source Simulation Project (eOSSP) was designed in this regard to initiate a radical change towards simulation-based product development, and to be a key driver for faster and more cost-effective development of better products in all industries. The eOSSP builds upon the finite-element code evolution-fea (e-fea) – which emerged from the AK32 Task Force 'Engine Simulation' of German car makers Audi, BMW, Daimler, Porsche and VW – and upon an open source business model.

"The promise of the eOSSP is lower cost, better quality, higher reliability, greater flexibility, and an end to predatory vendor lock-in!"

For more information:
payerconsulting GmbH
www.payerconsulting.com

Major new release of ParticleWorks unveiled

Particleworks™
Particle-based simulation software for CAE

Version 7 incorporates customer-requested features to facilitate and speed up complex simulations

Particleworks is a mesh-less computational fluid dynamics (CFD) software that allows liquid flows and heat transfer phenomena to be simulated using a particle-based method, or Moving Particle Simulation (MPS).

MPS solves the Navier-Stokes equations with a Lagrangian method and instead of discretizing the fluid domain with a grid, it uses particles, or calculation points, to populate the fluid volume. This mesh-free approach speeds up set-up and improves the stability of the simulation compared to finite volume CFD, particularly when dealing with liquid flows in complex geometries and when simulating moving boundaries.

There is a rather long list of industrial applications that are complex to approach using Finite Volume CFD, which become easy to configure and fast to run with Particleworks. Some of the most important ones include: simulation of forced lubrication and splash lubrication of transmissions systems, lubrication of bearings and engines, gradeability analysis and oil sump aeration, cooling processes of IC engine pistons, and cooling of electric-motors.

Particleworks 7 is a major release which introduces a large number of new features and models.

A new graphical user interface (GUI) ensures full integration between the GUI and the pre-processor / solver / post-processor and enables the use of Python API to script the entire simulation process, which offers significant advantages in terms of time reduction and automation of configuration, execution and post -processing of simulations.

Development of version 7's solver focused on heat transfer and multi-phase capabilities. Particleworks 7 has a new solver for heat conduction in solids and for the calculation of fluid-solid conjugate heat transfer. There are also improvements and increased flexibility for predicting the heat transfer coefficient, which is crucial for applications such as the cooling of pistons and e-motors.

In terms of multi-phase simulation, Particleworks 7 has a new built-in finite volume airflow solver, which makes the simulation of two-phase liquid-gas more flexible and faster than in the previous version.

The new version also integrates with third-party software via functional mock-up interface (FMI) a free standard for dynamic model exchange and co-simulation.

In addition, and even more importantly, a new product, the visualization option, allows you to translate Particleworks results into a format that can be read by computer graphics and virtual reality tools to create realistic renderings and for a better understanding of simulation results. This article will examine these features in greater detail.

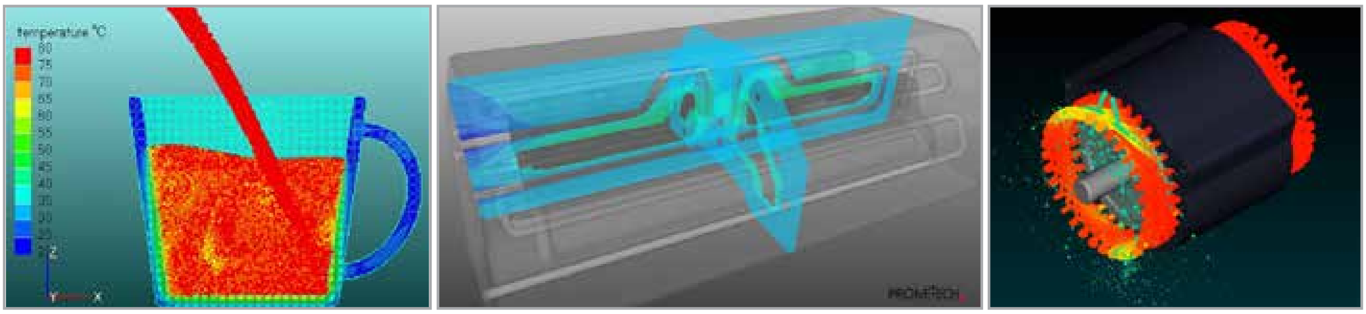


Fig. 1 – Three examples of the calculation of conjugate heat transfer using the new solver: the heating of a cup in contact with hot coffee, temperature distribution in a heat exchanger, and the cooling of e-motor windings.

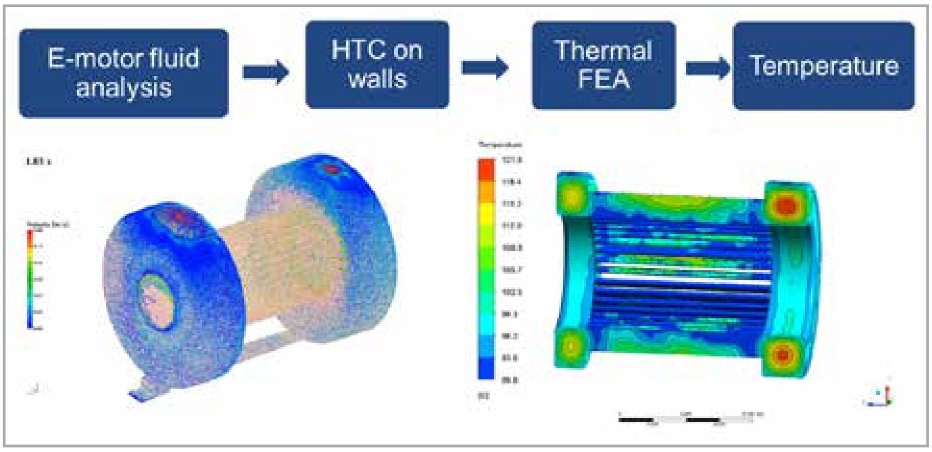


Fig. 2 – Simulation process commonly applied to e-motors and piston cooling analysis with HTC prediction from Particleworks and temperature calculation with FEA

Airflow modeling and Two-Phase flow

Particleworks 7 has a new embedded airflow solver that is fully coupled with the liquid MPS solver. The air solver is a finite volume CFD based on a Cartesian grid, which means that, as with the heat conduction solver, mesh generation is not required and that Particleworks automatically creates a Cartesian grid based on the wall spacing definition. By coupling the airflow solver with the liquid MPS solver, the gas and liquid phases can be simulated simultaneously, allowing for the reciprocal influence of the two fluids (multi-phase calculation with momentum transfer).

Heat Transfer

As mentioned, Version 7 introduces a new solver for conjugate heat transfer between fluids and solids that uses the finite volume method for the solid part of the simulation, while the standard particle-based MPS method calculates the heat transfer on the fluid side. The new finite volume method does not require the meshing of the solid bodies. Particleworks automatically generates Voxels to calculate heat conduction in solid bodies. The dimension of a Voxel is equal to the spacing defined by the user for the boundary condition definitions of the wall.

This method eliminates the need to generate solid particles to solve heat conduction, while the Voxel dimensions can be different from the particle size.

For some applications, when the time scales of fluids and solids are very different and transient thermal phenomena are not of interest, the best way to predict fluid-solid heat transfer and temperature is to decouple the simulation of transient fluid from the thermal conduction analysis. In this case, Particleworks predicts the fluid flow and the heat transfer coefficient (HTC) on the solid walls and maps the HTC distribution onto a Finite Element Analysis (FEA) model. Heat conduction analysis under steady-state conditions can calculate the temperature distribution of the solid in a few minutes.

Particleworks' primary role in this process is accurate HTC map prediction, and version 7 includes a user-defined HTC correlation. The advantage of this new feature is that the user can define the HTC correlation based on local flow conditions. In this way, the HTC prediction can adapt to the flow characteristics even in complex systems, such as an electric motor, where the flow behaves differently in different areas (flow parallel to the wall, impinging jet, pipe flow or other, depending on the region of the electric motor).

GUI

The introduction of the viewer API and Python API are among the main new features of the GUI. In the previous version of Particleworks, scripts could only be created for the solution and post-processing phases. With the new version, the user can script and automate the entire process from project creation, model definition, physical properties and numerical set-up, to solution and post-processing. Automation of the simulation process via C++ or Python drastically reduces manual effort and time for repetitive operations, e.g. for comparing multiple designs or the operating conditions of the same system. It also paves the way for process integration and design optimization to be applied to Particleworks models.

Some seemingly minor, but actually highly useful new features are also available in the new user interface. To mention just a few, in Particleworks 7 you can compare scenes to detect the configuration differences between different simulations and you can copy and paste objects, and define periodic movements for a faster set-up. In addition, there is a new task manager with improved usability.

In terms of solution monitoring, the user can now define live plots of physical and numerical quantities that are displayed in the interface and automatically updated during the solution. This makes it possible, for example, to monitor the behavior and stability of the solution and to obtain an immediate idea of trends in physical quantities such as torque or thermal balance.

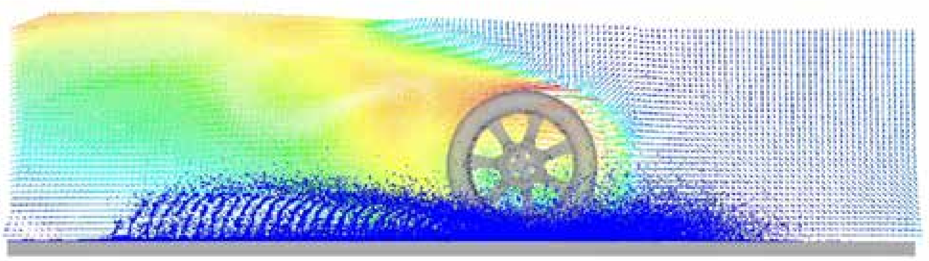


Fig. 3 – Water spray and aerodynamic analysis. Particleworks 7 calculates both airflow and water flow caused by the moving tire.

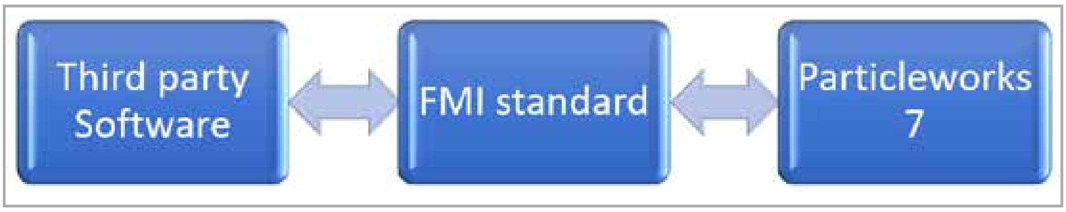


Fig. 3 – Particleworks 7 supports co-simulation with third-party software via FMI.

Moreover, Particleworks 7 adds the multi-view feature, which allows multiple scenes to be displayed in a single window. The user can display different views of the same simulation at the same time, show different variables for different views, or visualize the results of two or more projects or scenes to compare them.
These and other new features have been developed and implemented using customers' feedback with the aim of making their work easier and more efficient.

Pairing with third-party software

Particleworks 7 can dynamically interact and co-simulate with a wide range of third-party software via functional mock-up interface (https://fmi-standard.org/). The list of tools that support FMI, available on the FMI standard web-site (https://fmi-standard.org/tools/), is quite long.

One of the most interesting pairings and co-simulations is with Adams, a well-known multi-body tool. An important application is wading simulation. A Particleworks-Adams co-simulation can account for the dynamic effects of the vehicle and the vehicle-water interaction at the same time. Co-simulation allows you to calculate vehicle motion and water splashing simultaneously, taking into account the effect of water forces on vehicle dynamics.

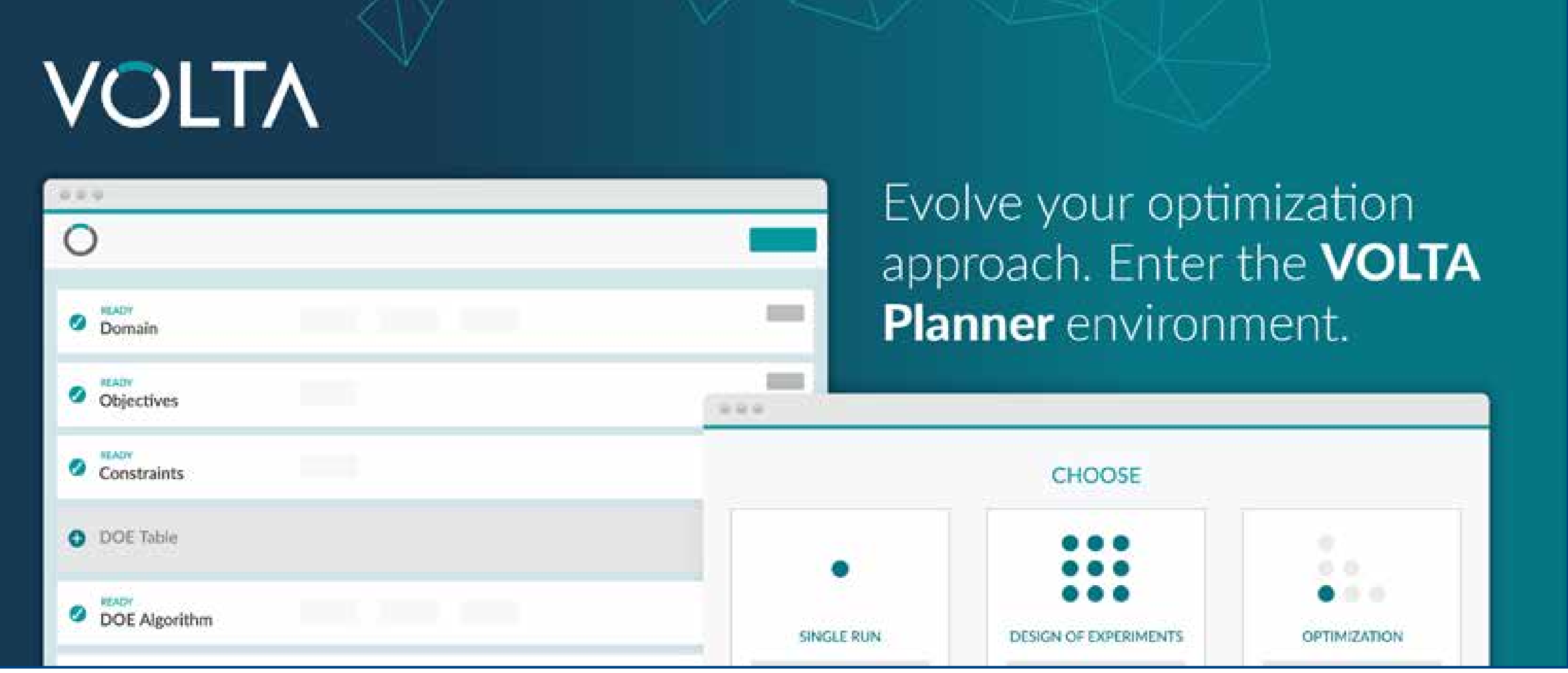
A special case is the coupling of Particleworks with Ansys Mechanical and Ansys Fluent. A dedicated Particleworks-for-Ansys interface was already available for version 6 and its development has produced a new version compatible with Particleworks 7 with new features. The Particleworks-for-Ansys interface allows the user to automatically transfer data from Particleworks to Ansys Mechanical for constant and transient stress and thermal analysis, and Ansys Fluent can transfer constant and transient airflow fields to Particleworks. This is an alternative way to measure the effect of airflow on liquids flows.

Visualization

An important way to understand and communicate the results of a simulation is a realistic rendering with the ability to "enter and view the results from the inside". This is possible in Particleworks 7 thanks to a dedicated plugin that converts its results into a graphical format, editable with computer graphic tools like Blender, Unity or others. You can see the effect in Fig. 5, where the water splashes calculated by Particleworks were rendered using a Virtual Reality tool. Pricing and availability and training for the new version are available upon request.

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

Fig. 5 – Rendering of a wading simulation via Virtual reality.



Design optimization gets easier and faster with ESTECO VOLTA Planner

The product development process in an enterprise context requires companies to promote collaboration among teams and departments. For instance, in the design stage, more engineers need to work on the same project and share their knowledge to reach the final objective more effectively.
In such a scenario, a flexible environment that connects all parties is essential. With VOLTA, the collaborative web platform for simulation process and data management (SPDM) and design optimization, ESTECO provides companies with an innovative solution to bring people, processes, and technologies together.

In VOLTA, design engineers can benefit from state-of-the-art algorithms to perform design space exploration and optimization and share their studies with other experts. When applying this optimization-driven approach to look for alternatives to the product design, manually setting up multiple optimization strategies may be very time-consuming. In such a scenario, engineers can access a modular interface in VOLTA to apply several optimization strategies in a shorter time. VOLTA Planner is designed as a single environment where teams and engineers can easily collaborate. All players involved can quickly set up several design space exploration plans and make them available for further improvements. At the same time, other users can execute simulations as many times as needed without handling the workflow configuration. VOLTA Planner is specifically intended to let the engineer focus on the results rather than on the set-up.


VOLTA Planner

FOCUS ON THE ENGINEERING GOAL

With VOLTA Planner, process automation can be split from the optimization strategy.
Engineers can choose among preconfigured scenarios and set their objectives to build their DOE and optimization studies. All automation steps are managed by VOLTA, which also suggests the best algorithms to use. In this way, set-up becomes quicker and the user can save time to concentrate on the improvements to be achieved.

IMPROVED COLLABORATION

VOLTA Planner enables all teams involved to save and reuse multiple design exploration strategies for the same project. Design engineers from different departments can access and compare existing optimization plans that were previously configured by other colleagues, apply their changes, analyze, and share their results. This helps improve collaboration and cultivate enterprise simulation knowledge.

FAST AND EASY SET-UP

The software smoothly guides the user through the configuration of the design exploration studies. It is built on a user-friendly, modular interface to quickly create multiple strategies. The creation of optimization plans is faster, freeing the user's time to innovate.

To find out more about VOLTA, visit esteco.com and ask for a demo.

Improved blast furnace performance with material load optimization

Combining modeFRONTIER with Rocky DEM to design a better deflector saves up to 130 hours of computation time

The Arvedi Group approached the University of Trieste to find a solution to the uneven distribution of material inside the hopper of their blast furnace in Trieste, Italy. The university's Mechanical Engineering Department studied the problem and used modeFRONTIER to optimize the design of a new deflector ensuring better material distribution. Leveraging ESTECO's integration and process automation technology, they coupled modeFRONTIER with Rocky DEM software to accelerate the simulation process of the material distribution. Using the proprietary algorithms available in modeFRONTIER, they were also able to find the optimal design for a new deflector.

The challenge

The project concerned the process of loading coking coal and iron ore into the hopper. These different materials had formed piles and pitches, leading to a reduced efficiency of the plant. The uneven distribution of the material inside the hopper caused variations in the temperature profile, gas flow, and gas composition.

To solve this problem, modeFRONTIER was coupled with Rocky DEM to better understand the behavior of the materials and to optimize the design of the deflector. The modeFRONTIER integration also made it possible to meet the time constraints by reducing the computational time for each simulation.

The solution

This project was developed in two phases. The first phase concerned the calibration of the Rocky DEM parameters and the simulation of the loading of the hopper. The second phase consisted of optimizing the geometry of a new deflector for the loading process.

The discrete element method (DEM) parameters, such as static particle-particle friction and rolling resistance, were used as inputs to modeFRONTIER for the calibration process. The simulated material's angle of repose was used as an output.

For the optimization of the equipment, a Uniform Latin Hypercube sensitivity analysis ran 90 designs and identified the most important design variables. The engineers then optimized three different geometries, using these geometric variables as inputs. The outputs were based on the distribution of the material, which was calculated by virtually dividing the hopper into 12 sectors and

"With modeFRONTIER we could evaluate 1000 designs in just a few weeks"

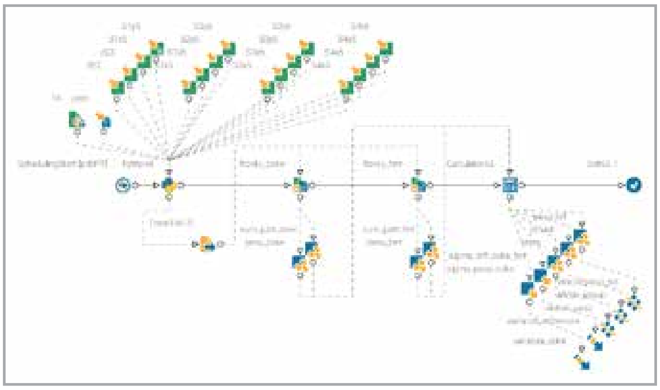


Fig. 1 - The workflow set up in modeFRONTIER to automate the simulation process in Rocky DEM and run the design optimization strategies.

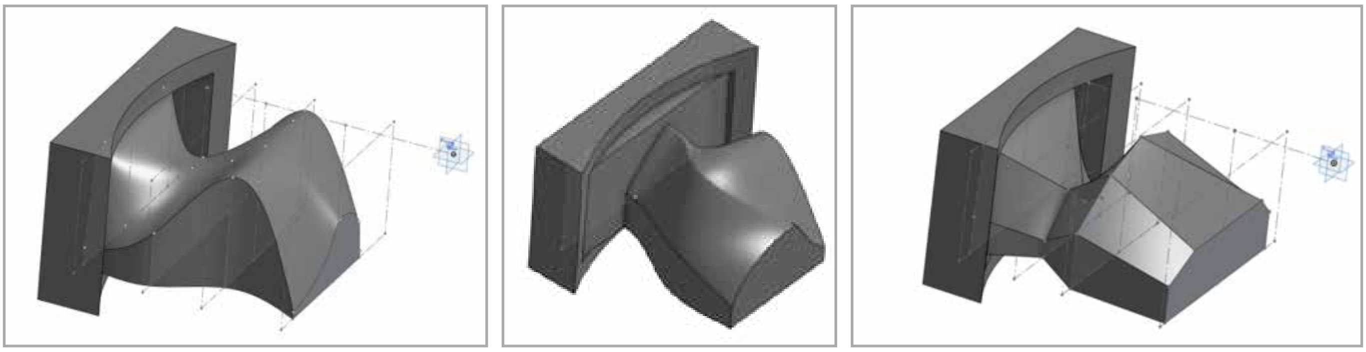


Fig. 2 - Three deflector designs were tested, using the pilOPT algorithm in modeFRONTIER.

performing a statistical analysis of the particles found in each one. These values were then used to define the two objectives and the optimization constraints.

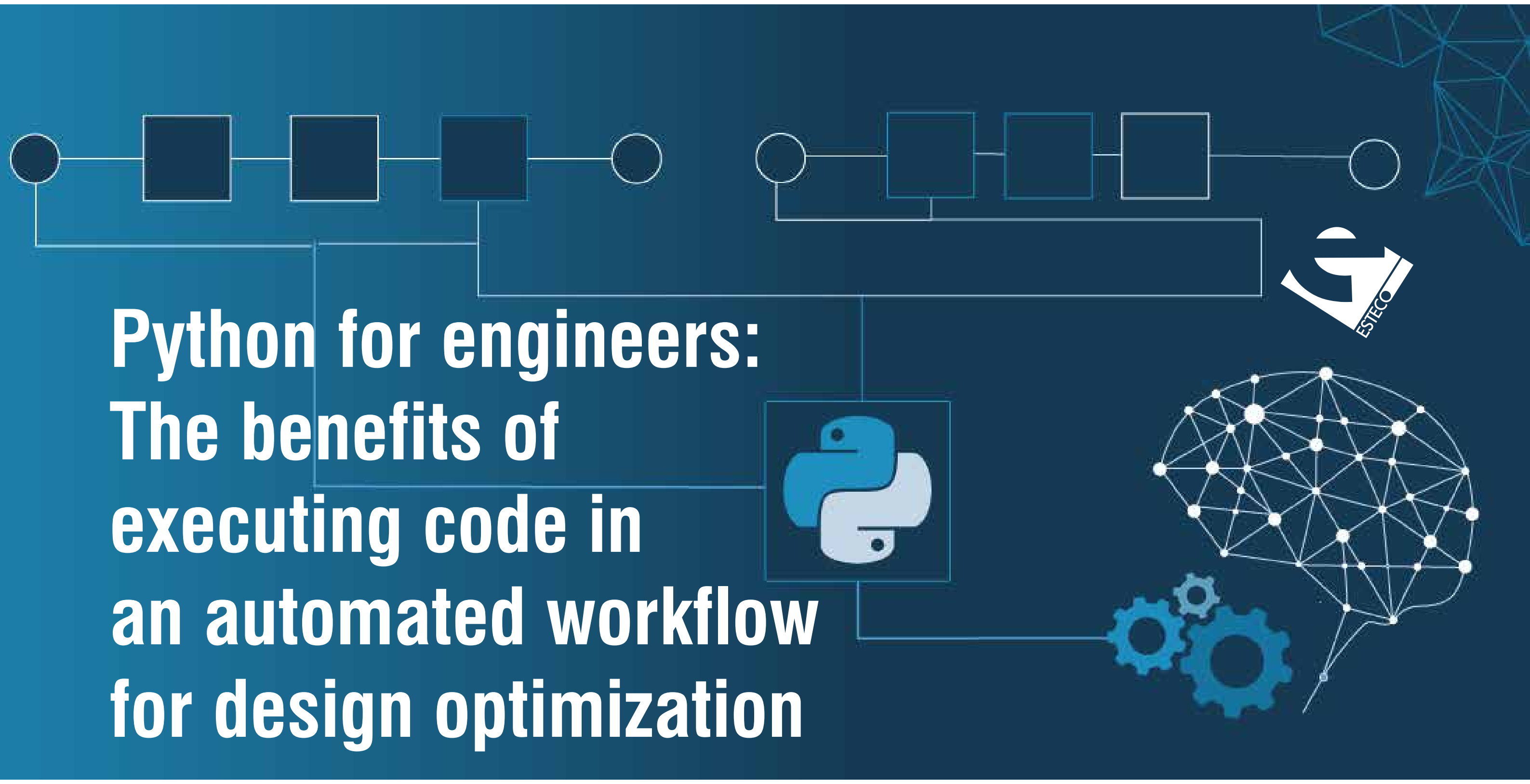
The department used ESTECO's proprietary pilOPT algorithm to run the three optimization studies. By using the autonomous mode, they were able to evaluate more than 1,000 designs in just a few weeks, without having to set any parameters and with considerable time advantages.

The benefits

With its user-friendly graphical user interface, modeFRONTIER helped automate the simulation process. Without modeFRONTIER, engineers would have had to manually modify the geometry of the deflector for each simulation, wasting a significant amount of time. With modeFRONTIER they were able to save up to 130 hours of computation time. Lastly, by automating the process, the design engineers could launch the optimization and avoid the painstaking process of manually combining the output from multiple applications.

> **About Arvedi Group**
> Founded in 1963, the Arvedi Group is one of Europe's most significant steelmaking realities, operating in the production of hot-rolled pickled, galvanized and pre-painted coils, carbon and stainless steel tubes and stainless steel precision-rolled strip.

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



Python for engineers: The benefits of executing code in an automated workflow for design optimization

Why Python meets engineering needs

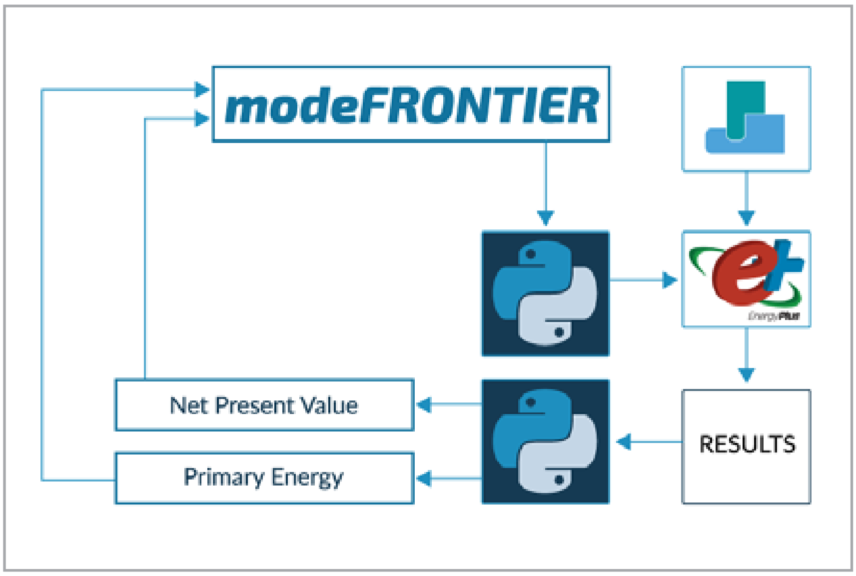
Nowadays, deep expertise in scripting and programming languages has become a valuable skill set for engineers and R&D professionals in general to perform process automation, data analysis, and visualization.

- Python, among other human-readable and open source programming languages, has gained popularity over the past decade due to its simplicity and flexibility:Programmers be they beginners or experts, need only focus on problem-solving rather than struggling with the nuances of programming languages
- Numerous scientific, engineering and web-related extension libraries, along with an enthusiastic community save time and provide valuable support in using the features of the program
- Python can automate everything, from submitting HTTP requests and interacting with APIs, to loading and managing databases
- Advanced machine learning capabilities enable engineers to elevate their data analysis efforts

Why Python combined with an optimization workflow helps to master engineering complexity

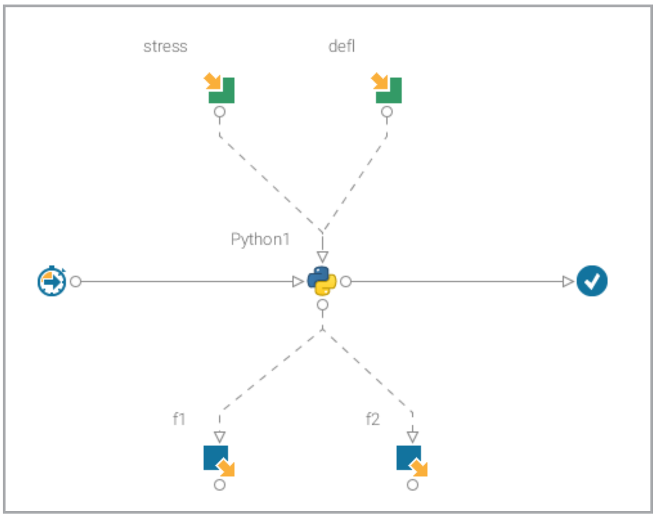
Simulation activities have mostly been conducted by engineers in the aerospace and automotive industries. They are able to use a variety of best-in-class software to perform analyses across many engineering disciplines and reduce development time and costs. Typically, numerical methods are required to solve complex engineering problems. In this context, Python has become a useful programming

language for the development of engineering applications and to facilitate numerical computing. In addition, the combination of scripting languages like Python with ESTECO process automation and integration technology makes it possible to create efficient pre- and post-processing workflows, easily execute the code, and manage very large datasets for effective exploration and analysis.
In addition, simulation has recently been gaining popularity in the architecture, engineering, and construction industry where innovative solutions must be balanced with sustainability and profitability in order to complete sophisticated buildings within tight deadlines and to high industry standards.



Typically, building performance modeling and simulation rely on complex models generated by different simulation tools with limited automation capabilities, making the configuration and execution of parametric studies a very time-consuming task. Thanks to ESTECO technology and its Python integration capabilities, researchers from the University of Trieste's Department of Engineering and Architecture were able to explore the effect of climate data on the energy efficiency of a refurbished building for social housing. The energy reduction interventions involved the application of internal insulation layers and the replacement of existing windows with more efficient ones.
The ESTECO modeFRONTIER software was used to guide the optimization strategy with the aim of minimizing primary energy and the net present value of the refurbishment investment for three different weather profiles.

Starting from a script implemented in Python using the eppy library, the code modified the characteristics of the building model, ran the simulation in EnergyPlus, and calculated the investment costs and the net present value. By easily importing their own Python script into the modeFRONTIER workflow, researchers were able to improve the process automation and apply multi-objective optimization algorithms to find the best designs with minimal primary energy consumption and refurbishment investments.



Data science and engineering teams can easily interact with modeFRONTIER via a Python script node and then create a custom workflow to meet their needs for a specific simulation design process:

- Integrate their own Python interpreter in one click
- Use high-quality written code and insert insightful models into the modeFRONTIER workflow
- Automate specific solvers and guide the execution of simulation chains to perform design exploration and optimization studies

Python integration is actually part of ESTECO's broad commitment to providing software solutions that maximize interoperability and best leverage your investment in simulation and modeling technologies. In fact, our leading integration and process automation technology helps you to manage all the logical steps of your engineering process from a single automated workflow.

modeFRONTIER, process automation and design optimization software, makes it possible to directly integrate and run many third-party CAD and CAE tools to perform design exploration and optimization studies. In addition to commercial engineering solvers, our wizard-style tools probably provide guidance to easily integrate any in-house solvers and your favorite programming languages such as Python.

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 VOLTA, the collaborative web platform for Simulation Process and Data Management and design optimization, and modeFRONTIER, the comprehensive solution for process automation and optimization in the engineering design process.

TGSim Plus™: a real-time dynamic simulator of gas turbines

First full-product customer approves the simulator's real-time behavior and passing of accuracy requirement tests

By Attilio Brighenti, Davide Duranti, and Debora Quintabà
S.A.T.E.

TGSim Plus™ is a software application developed by for the real-time dynamic simulation of a gas turbine (GT) power generation plant, including auxiliary gas and liquid flow systems (AUX), in order to verify and validate the GT control system (GTCS) by means of hardware in the loop (HIL) simulations, before its commissioning (Fig. 1). This software, which is based on 's extensive background in the simulation of turbomachinery [1, 2, 3, 4, 5, 6, 7], has recently been licensed to Doosan Heavy Industries and Construction (DHIC, South Korea) for use in their development of a new large class gas turbine [8].

This advanced engineering tool is useful for the design and analysis of GT systems, and to verify the tuning and logic of the control system, and the reaction of the diagnostic systems to emergency situations or failure conditions that could damage the machinery. It provides insight into system and component performance, investigating the real-time plant's response to control actions and enabling the identification of the most effective design and control settings to optimize transients and steady-state operations.

TGSim Plus™ simulates the thermodynamic behavior of the GT's main process based on 0-D lumped elements [9, 10] such as the compressor, combustor, expander, manifolds, valves and restrictions, including the variation of the compressor maps with the inlet guide vanes angle (IGV), bleed management and GT electric starter motor-generator model.

The real-time model has been integrated into a simulation suite that allows complete system models to be built based on high-level block libraries, and manages the pre-processing and loading of data, and

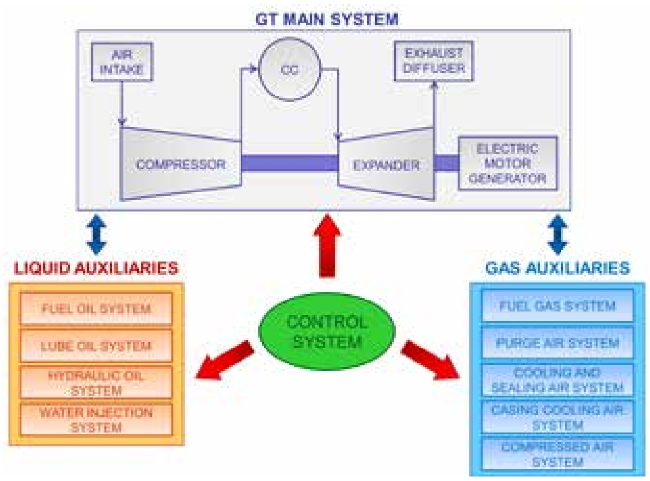


Fig. 1 - GT main system, auxiliaries, and base control system interactions. (Published with permission from "Hydrocarbon Engineering" [6])

the simulation of dynamic and steady-state conditions, such as start-up, shutdown, and load rejection. The DHIC-specific version was tested and delivered for a DS1006 multiprocessor architecture by dSPACE GmbH (DE), operating at a millisecond time scale.

Program description

TGSim Plus™ runs in the MathWorks™ MATLAB®/Simulink® environment, a powerful and qualified mathematical and systems simulation suite. The program includes a user-friendly Graphical User Interface (GUI), based on buttons and menu-driven commands.

A set of libraries of customized, ready-to-use, proprietary SATE blocks allows a model to be built by selecting and connecting the component blocks. This makes it highly flexible to use, allowing the model's complexity to be increased gradually, as and when necessary, based on the required behavioral details and runtime constraints.

TGSim Plus™ can manage two types of model: real-time online models for HIL simulations that interface with the HIL platform; and standalone offline models for GT design and analysis, verification of control strategies (through a partial software implementation of GTCS), and simulations to study or validate the physical system against test or reference data based on the simulated control system procedures to manage the GT and AUX systems.

The following relevant aspects were implemented for DHIC:

- Compressor thermodynamics, with interpolation of compressor maps – that allows the compressor to be modeled either as a single-map compressor or as a multiple-map compressor, subject to the availability of performance maps of the entire machine, or of separate sets of stages, or split into separate stages. The calculation of the operating point and the management of the extracted mass flow rates for bleeding and cooling are also included.
- Combustion chamber thermodynamics, modelled as a two-zone combustor that simulates the combustion process by considering stoichiometric combustion of fuel (gas, liquid or both) with dry air, and the mixing process of the stoichiometric combustion products with the excess inert air and water vapor that enters as humid air stream.
- Expander thermodynamics, including interpolation of expander maps, calculation of active torque and the heating of cooling streams due to heat loss from the equipment mass along the bleed lines from the compressor to the expander.
- Operations of the electric motor-generator, considering two mutually-exclusive functional modes, activated according to the GT's operative phase: electric motor (EM) mode, where the static frequency converter (SFC) and its internal control drive the variable-speed electric motor; and electric generator (EG) mode, based on a synchronous machine model, that simulates the GT system load before and after connection to the grid.

Procedure outline

The user defines the input data for the simulation in an editable MS Excel® file, which provides the settings for the Simulink® model. The data file can be edited according to the model assembled for the specific plant in the one-off model build-up phase.

For online simulations, thus specific to a given HIL architecture and application, the simulation results are displayed in the HIL platform's GUI.

For offline simulation results, TGSim Plus™ provides a variety of output graphs including time histories of all state variables and key operational and thermodynamic properties, which can also be printed or exported to other environments.

Typical results

The typical results of a transient simulation from start-up to powering and shutdown using TGSim Plus™ are shown in Fig. 2 and are compared to the reference data obtained with a GT design tool [11] by DHIC.

The results are shown in normalized form in relation to the values of the respective variables in the component considered, at full-speed full-load (FSFL). The temperatures are normalized as absolute value ratios [K/K].

The gas turbine system was simulated from the combustion ignition up to the full-speed no-load condition (FSNL), and then from FSNL to FSFL, until it reached a steady-state condition. The load was then reduced back to the FSNL condition, which was followed by shutdown. To enable the reference data to be compared with the TGSim Plus™ results using offline simulation, the reference signals for speed (as a set point), load power, and IGV angle were used as "exogenous inputs" for the model. In TGSim Plus™ the gas turbine operates with dynamic speed control for the entire simulation, subject to all other

Description	Deviation
Compressor intake mass flow rate	–0.28 %
Combustor exhaust temperature	–1.13 %
Combustor exhaust pressure	–1.96 %
Expander exhaust temperature	+0.20 %

Table 1 – Percent deviations between reference data and the TGSim Plus™ results at FSFL. Note that temperature deviations are based on a normalization of temperature values in [K].

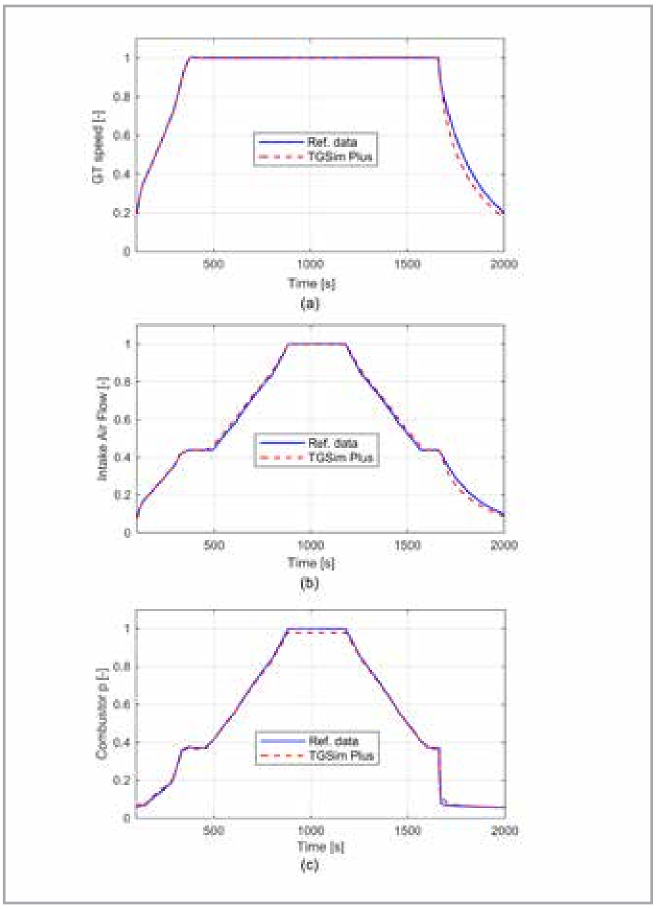


Fig. 2 - Results of start-up and shutdown simulation. Comparison between reference data (solid blue line) and TGSim Plus™ results (dashed red line). (Published with permission from the EVI-GTI Conference 2019 [5] and the "International Journal of Turbomachinery, Propulsion and Power" [7])
(a) Normalized rotational speed,
(b) Normalized compressor intake mass flow rate,
(c) Normalized combustor exhaust pressure.

dynamics: thermal, fluid, rotational and other controllers. On the opposite, in the DHIC design tool the shaft speed is an "exogenous input" to the model.

The graphs in Fig. 2 show that TGSim Plus™ matched the reference data quite well, both during transient phases and under steady-state conditions. Table 1 shows the percent deviations at FSFL for the compressor intake mass flow rate, combustor exhaust temperature, combustor exhaust pressure, and expander exhaust temperature.

The main differences between the reference data and the TGSim Plus™ results occur in the transient phases. For the rotational speed and the compressor intake flow the differences are more relevant during the shutdown, as shown in Fig. 2 (a) and (b). The differences during the GT shutdown phase were caused by the different flows in the cooling and bleed lines in the reference data and the TGSim Plus™ simulations. These were a result of differences in the cooling and bleed control actions and of different estimates of the flow coefficients in the relative lines. These different flows have an evident impact on the balances of the various volumes and therefore on the trend of some variables. In support of this argument, it was shown that by modifying the head loss parameters (admittances) of the cooling and bleed piping and some parameters of the control logic, the simulated behavior was more consistent with the reference behavior, confirming the importance of this data and of tuning the control system.

Further details on transient simulation analysis using TGSim Plus™ are available in past publications [4, 5, 6, 7].

Conclusions

SATE's R&D activity in GT modelling and software implementation work have resulted in a powerful tool to simulate complete GT systems. TGSim Plus™ can be used both in offline mode, during the GT design phase to optimize and verify the control strategies, relying on integrated control functions and library blocks; and in online real-time mode to verify the control system and tests before coupling takes place with the actual machinery.

The differences between the reference data and the TGSim Plus™ results in the validation tests of the transient simulations fell within the accuracy requirements by the first full-product customer, namely DHIC (Doosan Heavy Industries and Construction, South Korea), which approved the tests performed and the real-time behavior of the simulator.

For more information visit:
www.sate-italy.com

References

[1] Brighenti, A., Bernocchi, A., Martini, F. et al.: "Simulating Snøhvit compressors: Part One", LNG industry, Palladian Publications: Farnham, Surrey, UK, 2007, Autumn, pp. 47–52.

[2] Brighenti, A., Bernocchi, A., Martini, F. et al.: "Simulating Snøhvit compressors: Part Two", LNG industry, Palladian Publications: Farnham, Surrey, UK, 2008, Spring, pp. 89-94.

[3] Brighenti A., Bondi S., Fioravanti L. et al.: "Simulator of the Oregen™ system for Waste Heat Recovery from Gas Turbines", 32nd CAE Conference, Parma (Italy) 17-18 Oct. 2016.

[4] Brighenti, A., Duranti, D., Quintabà, D.: "Real-Time simulation of Gas Turbine systems", 35th International CAE Conference, Vicenza (Italy) 28-29 Oct. 2019.

[5] Brighenti, A., Duranti, D., Quintabà, D.: "TGSim Plus™ – Real-Time Dynamic Simulation Suite of Gas Turbine Systems for the Matlab®/ Simulink® Environment", The 9th EVI-GTI International Gas Turbine Instrumentation Conference, Graz (Austria) 20-21 Nov. 2019.

[6] Brighenti, A., Duranti, D., Quintabà, D.: "Dynamic simulation", Hydrocarbon Engineering, Palladian Publications: Farnham, Surrey, UK, 2019, 24, 65–68.

[7] Brighenti, A., Duranti, D., Quintabà, D.: "TGSim Plus™ – Real-Time Dynamic Simulation Suite of Gas Turbine Systems for the MATLAB®/ Simulink® Environment", Int. J. Turbomach. Propuls. Power 2020, 5, 24.

[8] https://www.power-eng.com/2020/01/03/doosan-heavy-industries-deploying-first-large-class-gas-turbine-for-korean-demonstration-project/#gref

[9] Atherton, D.P., Borne, P. "Concise Encyclopedia of Modelling and Simulation", 1st Ed., Pergamon, 1991.

[10] Luyben W.L., "Process Modeling, Simulation, and Control for Chemical Engineers", 2nd Ed., McGraw-Hill, 1989.

[11] "Numerical Propulsion System Simulation (NPSS)", https://www.swri.org/consortia/numerical-propulsion-system-simulation-npss, accessed 25 June 2019.