ENGINSOFT

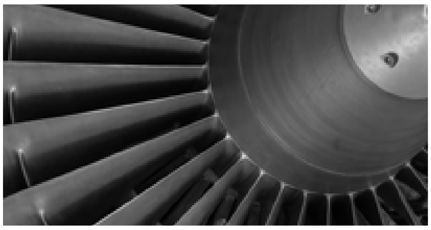
The Simulation Based Engineering & Sciences Magazine

Newsletter

Year 16 n°2

Summer 2019

ANSYS Solutions simplify additive manufacturing the example of a drone



The design optimization of a small axial turbine with millions of configurations

Using simulation and validation to build trust in new tools

Two-step approach to numerical simulation of fire and smoke propagation

CRISR

Centre for Interdisciplinary Research on Critical Infrastructure Security and Resilience

SUMMER SCHOOL

CyPhy2019: HANDS ON CYBER-PHYSICAL THREATS TO CRITICAL INFRASTRUCTURES

https://sites.google.com/unisalento.it/summerschoolcrisr2019 CRISRUnisalento@gmail.com

TOPICS

Theoretical lectures and practical use cases on the following aspects

- Concept of operations
- Simulations
- Physical and cyber risk analysis, prevention, detection, response and mitigation
- Evolving and hybrid threats
- Practical case studies on energy Infrastructures, transportation infrastructures, cybersecurity and their interrelations

DETAILS

For Whom: 'Next generation' security managers and young scholars

When: Sept. 25-28, 2019

Where: Convento degli Olivetani, Lecce, Italy.

How much: 1000 Euros.
Scholarships will be available

Why: it could be your next career step!

ORGANIZED BY:

UNIVERSITÀ DEL SALENTO

Department of Enginering for Innovation

SPONSORED BY:

snam

IN COLLABORATION WITH:

aipsa Associazione Italiana Professionisti Security Aziendale

EISAC.it

Con la partecipazione di ENEA

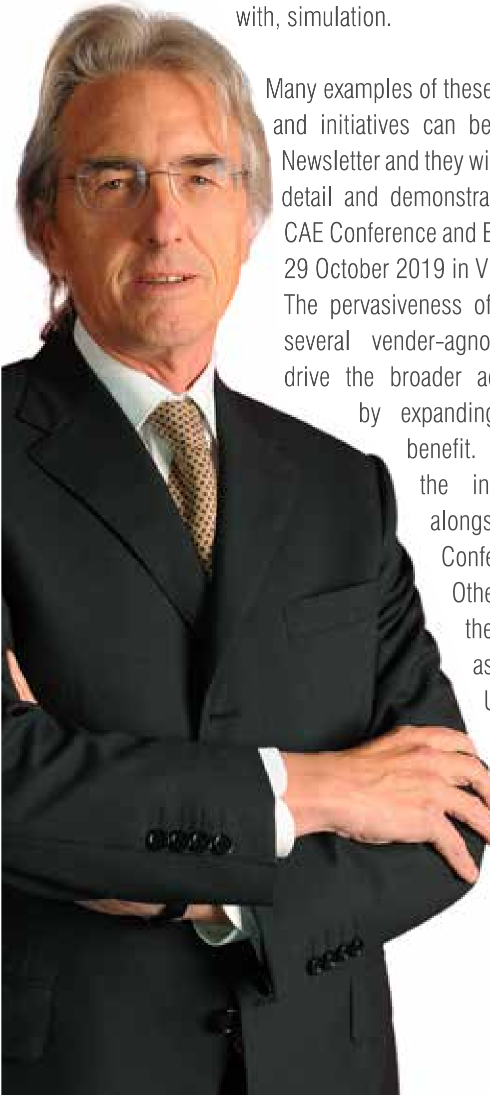
HS HOMELAND SECURITY MASTER

Contacts: sara.quarta@unisalento.it (+39 0832 299015)

Flash

Simulation based engineering and science (SBES) continues pervading all areas of industry, allowing the prediction and study of product and system performance from inception to end of life, for individual components and complex systems with multiple physical attributes and behaviors alike. Since it enables the study of phenomena and systems that are too dangerous, too expensive or too difficult to conduct actual experiments on, it has made powerful inroads in every industrial sector and is being used to solve some of the most pressing issues of our time, such as environmental protection, climate impact, food security and so forth. This means that simulation is being viewed by governments as fundamental to national competitiveness and economic wellbeing. Industry and business meanwhile remain focused on its increasing role in cutting costs, shortening time to market, and improving performance, reliability and energy efficiency.

Developments in ICT technologies, such as supercomputing processing power, high-speed networking, cloud-based solutions for processing and data storage, and new software licensing models have begun expanding its accessibility to small and medium-sized enterprises too – part of the so-called democratization of SBES, extending its use to engineers and users not expert in, or even familiar with, simulation.



Many examples of these and other SBES applications and initiatives can be found in this issue of the Newsletter and they will also be addressed in greater detail and demonstrated at the 35th International CAE Conference and Exhibition, to be held from 28-29 October 2019 in Vicenza, Italy.

The pervasiveness of SBES is also the result of several vender-agnostic initiatives seeking to drive the broader adoption of simulation tools by expanding their use and business benefit. The launch this year of the independent CAE Exhibition alongside the International CAE Conference is one example of this. Others are the launch in 2016 of the ASSESS initiative (www.assessinitiative.com) in the USA (ASSESS stands for the Analysis, Simulation, and Systems Engineering Software Strategies), which hosts an annual CAE gathering with the vision of simplifying access to simulation particularly for SMEs), and the launch in 2018 of Rev-Sim.org (revolutioninsimulation.org), which describes itself as "the online ecosystem for the simulation revolution". Yet other ongoing initiatives are spearheaded by the non-profit international association NAFEMS, who are also strong participants in the International CAE Conference and Exhibition.

Trends within the simulation industry itself are also fueling the increasing pervasiveness. The most significant of these is the digital twin which is increasingly being seen as the way to close the loop between the product design intentions and their real-world performance over the product or system's life cycle. However, the digital models need to be realistic and accurate enough to replicate the Multiphysics complexities the product or system experiences. This information can be used to shorten product development cycles, increase performance and reduce costs. These and other aspects of the deployment of digital twins, such as treatment and protection of the data they generate as a core business asset, will be further discussed at the Digital Twin Round Table event at the International CAE Conference and Exhibition.

Yet, this increasing pervasiveness of simulation is also adding tremendous impetus to an enduring and increasingly pressing challenge – that of creating a sufficient supply of simulation-skilled engineers. Graduating students with multi-disciplinary "classical" engineering education, who also have knowledge of and experience with high-performance computing, mathematical modelling, numerical simulation, artificial intelligence and more, are desperately required for this simulation-infused world. Even the "democratization" of CAE requires the ongoing services of experts to create, test and simulate the usability of the simplified tools, and the reliability and accuracy of the output they generate. The hard truth is that the number of simulation experts available to provide the knowledge transfer is highly limited. Some of this can be addressed by initiatives such as the CAE Conference's annual Poster Awards competition (www.caeconference.com/poster.html) which awards the creative use of CAE technology among university students around the world. But perhaps a faster, short-term solution is to skill up experienced "traditional" engineers through initiatives such as the Competence Centers in Italy (see page X of the Newsletter) or by attending focused, high-profile events like the International CAE Conference and Exhibition (by the way, interest is enormous, but there are still some opportunities to exhibit – contact the organizers at info@caeconference.com for more information). EnginSoft will participate substantially. We count on seeing you there!

Stefano Odorizzi,
Editor in chief

Contents

Case Studies

Software Updates

Events

PAST ISSUES at
www.enginsoft.com/magazine

Newsletter EnginSoft
Year 16 n°2 - Summer 2019

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

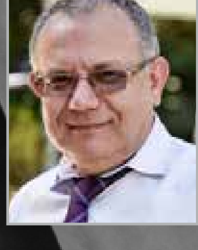
PRINTING
Grafiche Dalpiaz - Trento

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

Expendable, miniature, low-cost turbojet engines are often equipped with axial turbines. These turbines usually have a low-pressure ratio, a simple design of the blades and vanes, and a relatively low performance. In this article, we show that the main turbine characteristics, such as efficiency and exit flow angle, can be sufficiently improved using parametric optimization. Using a fast code for mean-line turbine design in modeFRONTIER's optimization environment allowed us to check about two million configurations and to determine the most important design parameters.

The design optimization of a small axial turbine with millions of configurations

The case for computerized optimization over manual design interventions



By Savely Khosid
Rafael - Advanced Defense Systems Ltd
Technion - Faculty of Aerospace Engineering, Haifa, Israel

Introduction

Increasing worldwide interest in expendable, low-cost turbojet engines for both civil and military purposes has led to the design and production of many miniature turbojet engines by companies and groups of enthusiasts in Germany, the UK, Spain, Taiwan, Denmark, Australia, the Netherlands, the USA, Russia, Japan, Israel and other countries [1]. These engines are usually equipped with a small low-pressure turbocharged centrifugal compressor driven by a low-pressure ratio, one-stage, axial turbine. Their engine performance is also quite low, with the specific fuel consumption (SFC) being in the range of 1.3-2.0 kilograms of fuel per kilogram of force (kg/kgf).

The large relative aerodynamic losses due to the small size of the components are responsible for the low performance of these miniature turbojet engines. On the other hand, the main components of these engines are usually not optimized because of the low cost of the engines and of their design.

This article describes a simple method to optimize the performance of a small axial turbine during the preliminary design stage by using a fast mean-line design code with the modeFRONTIER optimization software and a well-established genetic algorithm

Mean-line axial-flow turbine analysis

The layout of a miniature turbojet engine is shown schematically in Figure 1. Its main elements are a centrifugal compressor, a combustion chamber, an axial power turbine, and a converging propelling nozzle. A compressor increases the air pressure and pushes the air towards the combustion chamber, where the air temperature raises due to fuel

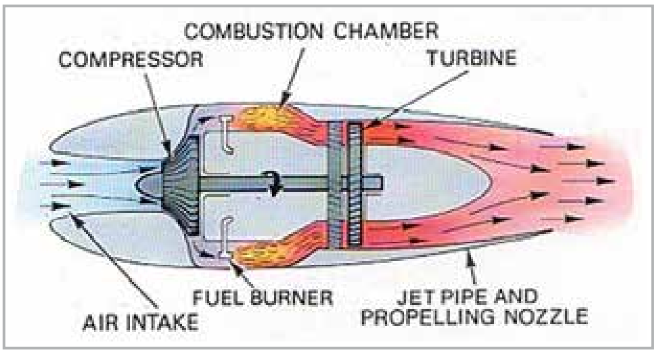


Fig. 1 - Miniature turbojet engine layout

combustion. These hot gases in the turbine produce the power to drive the compressor. They are then expelled through the propelling nozzle to produce the engine thrust.

The main objectives of a turbine's design are to convert the kinetic energy of the gas into the shaft power with maximum efficiency and to provide an axial non-swirling exit of the gas into the propelling nozzle. Additional objectives are withstanding the mechanical and thermal stresses, a necessary safety margin from resonance conditions, manufacturability, smooth blade profiles,

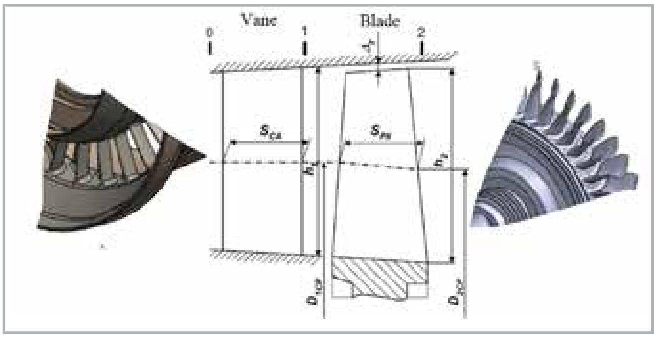


Fig. 2 - Axial turbine schematic and construction

etc. The power turbine in a miniature turbojet engine is usually a one-stage axial turbine, shown schematically in Figure 2. These turbines consist of static vanes that direct the flow into the rotating blades at the correct angle to provide momentum to the shaft. For

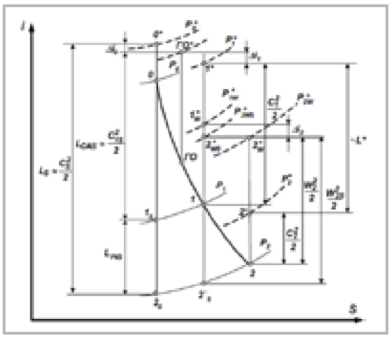


Fig. 3 - Expansion through the turbine stage

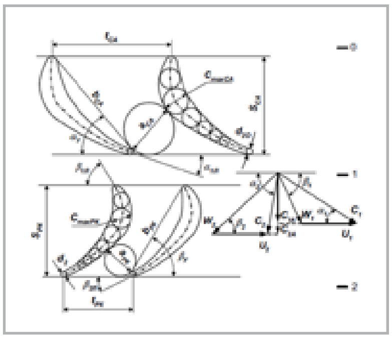


Fig. 4 - Vane and blade schematic

maximum turbine efficiency, the vanes and the blades must be perfectly aerodynamically matched. This is achieved by means of the careful design of the geometry of the vanes and the blades, known as mean-line analysis.

Turbine design is a multi-stage process where the first stage includes the analysis of mean-line performance, the setting of the turbine dimensions, the definition of the blade inlet-outlet angle, the selection of an aerodynamic loss model, and the efficiency calculation. The design of the vanes and blades is also subject to additional constraints, such as the profile smoothness. To obtain a high-quality preliminary turbine design, we used a two-dimensional calculation of the turbine parameters at five sections along the blade, instead of the usual one-dimensional, mean-line analysis at the median radius. Since a large number of input parameters affect the turbine's performance and the flow's exit angle, we used a multi-objective, multi-constraint and multi-variable optimization to improve the turbine's performance at this stage.

Axial turbine thermodynamic analysis is based on an enthalpy-entropy diagram of the process of gas expansion in the turbine stage, as shown in Figure 3. The relationships between the turbine parameters are determined by the geometry of the vanes and blades and of the velocity triangles, as presented in Figure 4. We varied these velocity triangles with each blade/vane section, while a profile stacking around a center of mass, shown in Figure

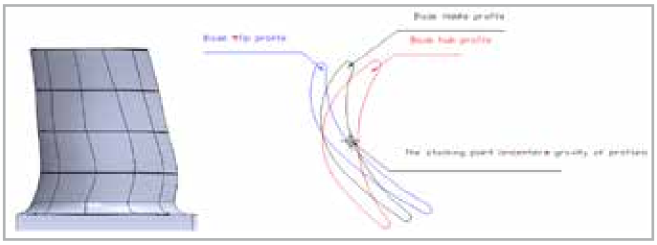


Fig. 5 - Blade sections and profiles stacking

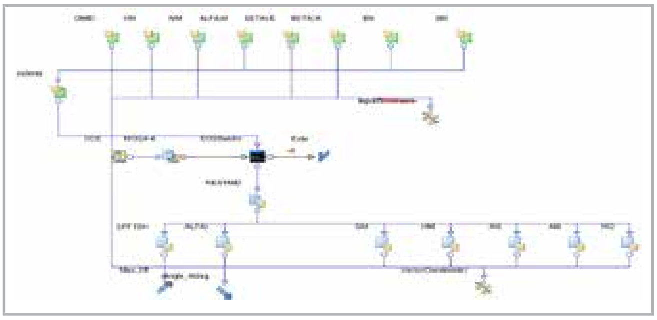


Fig. 6 - Optimization model

5, allowed us to reduce the blade/vane stresses and to achieve better turbine performance. Profile stacking produces a quasi-three-dimensional form in the vanes and blades, which generates a significant improvement compared to the one-dimensional design of the median radius.

The fast Fortran-compiled, in-house code for two-dimensional mean-line analysis of an axial-flow turbine written by the late R. Priampolsky was used for this project. The program input file includes a large number of parameters, such as turbine mass flow rate, stage inlet temperature and pressure, obtained at the outlet of the combustion chamber, outlet pressure, pressure recovery in the inlet duct, gas properties, rotational speed, the hub and tip radiuses of the vanes and blades, their numbers, radial clearance, hub leakage temperature, inlet angles, outlet velocity coefficients, preliminary estimates of velocity loss coefficients, and blade profile characteristics. In addition, the vane and blade inlet and outlet angles, and trailing edge thicknesses are provided for each of the five radial sections. All this data is provided in the constant textual tabular form that allows the optimization model to conveniently change all the input parameters.
Mean-line turbine analysis uses the appropriate losses model and conventional blade/vane shapes, and provides the user with the radial distribution of the velocity triangles, pressures, temperatures, densities, etc. Turbine power, friction losses, stress at the hub and efficiency are also calculated and written into the output ASCII file.
A reliable model of the energy losses in the flow path of an axial-flow turbine allows the accurate prediction of the turbine's characteristics in 1D and 2D mean-line analysis. As mentioned in [2], more than ten complex models are known today, together with dozens of equations calculating the different loss components. These models and loss equations have been obtained over the last 70 years by researchers around the world. The authors of [2] analyzed the results of experimental investigations of the profile losses for more than 170 non-swirling cascades of axial turbines with a constant height section. These experiments were performed by several gas turbine manufacturers from the former USSR for many axial-flow turbine blade profiles used in aircraft gas turbine engines. Based on this data, the accuracy of the five most popular loss models was compared and summarized in [3]. As was shown, the CIAM model of axial-flow turbine losses is the best of the five popular models for a wide range of turbines from the perspective of maximum error and standard deviation. Therefore, we used this model for the present analysis.

Optimization module

The code for a 2D axial turbine mean-line analysis is written in Fortran IV and compiled into an executable (exe) file. This code includes 19 subroutines including the CIAM empirical loss model and is able to treat both subsonic and supersonic flow conditions in the turbine. After initiation, the executable file reads the input file, performs the calculations and writes the results into a formatted output file in ASCII format.

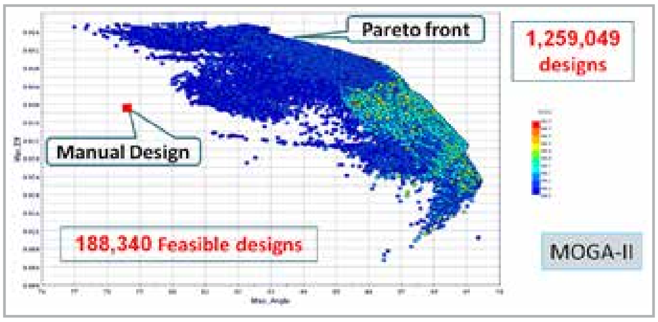


Fig. 7 - Optimization with N>1,000,000 designs

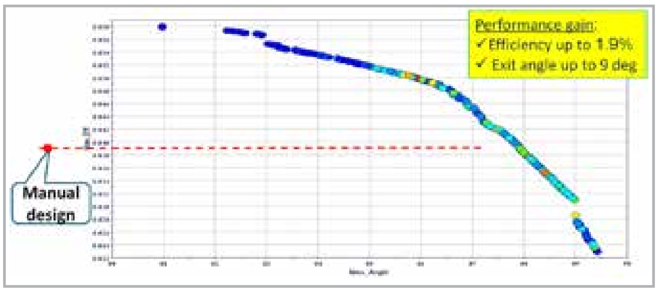


Fig. 8 - Pareto designs (first optimization)

The optimization of the turbine's performance was undertaken using the modeFRONTIER (mF) tool, developed by ESTECO. A general layout of the optimization model is shown in Figure 6. The mF workspace consists of five main blocks: the input parameters block, including the constraints; the design of experiments (DoE) block, where the initial (parent) population of parameters is randomly chosen; the optimizer block, where the initial parameters are changed; the execution block that produces the output file; the target block with the constraints on the output parameters. The input and output files are connected by the user-defined node that allows the user to perform command-line operations on the executable file.

In the present study, the multi-objective genetic algorithm (MOGA-II) was used to treat a large number of input parameters

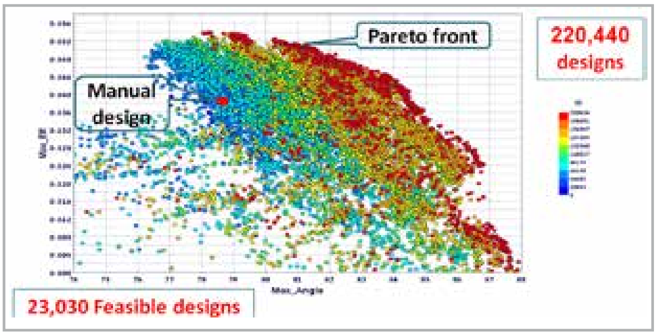


Fig. 9 - Optimization with limited blade angle of attack

and the numerous constraints on the output parameters. Two design objectives were defined: the turbine's thermodynamic efficiency, and the mean flow angle at the turbine's outlet. A minimal swirl of the outlet flow allows maximum thrust to be developed in the turbojet engine's propelling nozzle. Our model included 33 input variables and 20 output parameters, including turbine efficiency, shaft power and the blade stress estimation. The model under consideration works under the limitations of 139 constraints including 66 min/max constraints on the input variables, 38 constraints on the geometrical interactions, 15 constraints on the monotony of the radial vane/blade profiles and stacking, and the 2 design objectives namely the requirements for the mass flow rate and the turbine power. Among other limitations was the requirement for the number of vanes and blades to be a prime number to reduce the likelihood of harmful resonances. It should be noted that, at the preliminary stage of the manual analysis by a very experienced designer, this constraint prevented the attainment of an acceptable level of efficiency and was omitted. The limitation was returned and fulfilled by the computerized optimization process. Based on the large number of initial variables, and to obtain a representative variability in the initial (parent) population, 10,001 DoE sets of variables were formed using the Sobol deterministic DoE algorithm. The Sobol algorithm mimics a random choice to obtain a uniform sampling of the design space and to reduce the clustering of parameters. The output parameters were also limited by the defined power of the turbine and the degree of reaction at each section. A total of 18 constraints limited the output.

Because of the large dimension of the problem and the complexity of the constraints, the optimization was performed using the MOGA-II genetic algorithm. The elitism procedure of this algorithm, which preserves excellence, ensures that each new generation's performance is greater than the performance of its parent generation.

Results

At the first attempt, the optimization model was run through 126 generations to produce 1,259,049 designs. Because of the strict constraints on the output parameters, only 188,340 of the designs (about 15%) were feasible. Figure 7 clearly demonstrates the superiority of the computerized optimization over the manual design. Figure 8 shows the Pareto frontier of the best designs from the first optimization, which generated a 1.9% performance gain in efficiency and 9 degrees of the exit angle. The Pareto graph shows a few discontinuities that clearly demonstrate the non-linear nature of this problem and the advantage of using a genetic algorithm in this case. Despite the success of the first phase of the turbine design, a more rigorous analysis of the optimized designs presented in Figure 8 revealed unexpectedly large angles of attack (DBeta1) between the blades and the gas flow (up to 20 degrees). The problem, here, is that the turbine loss model has not been validated at such angles, so another limiting condition was added in the form |DBeta1|≤10deg. A further 220,440 configurations were calculated using the Non-dominated sorting genetic algorithm II (NSGA-II). According to mF [4], NSGA-II implements the crowding distance approach which guarantees the diversity and spread of the solutions on the Pareto front by estimating the

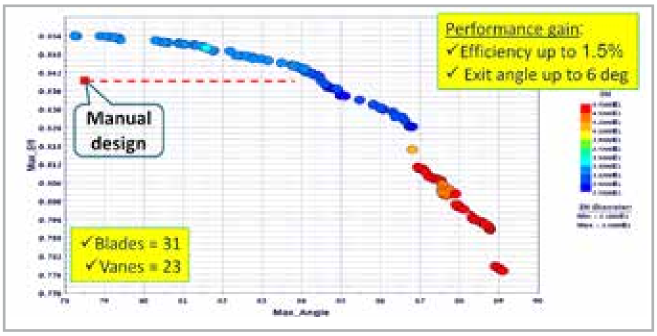


Fig. 10 - Pareto designs (second optimization)

density of the solutions in the objective space and guiding the selection process towards a uniform spread. The points belonging to the same front are sorted such that a higher ranking is given to the points located in the less populated regions of the front. Since this sorting demands additional calculations, we used the NSGA-II algorithm because of the smaller population. No advantage over the MOGA-II algorithm was observed.

After the addition of a constraint on the blade's angle of attack, only 23,030 feasible designs were obtained (about 10%) from this second optimization. Figure 9 shows that the best efficiency reached was 85.4%, which reduced to 85.2% when the exit angle was above 80 degrees (generally, the exit angle should be close to 90 degrees).

Figure 10 shows the Pareto frontier of the best designs from the second optimization. A prime number of vanes (23) and blades (31) was obtained, with a potential performance gain of up to 1.5% in efficiency and 6 degrees of the exit angle.

To maximize the efficiency of the optimized turbine, another 546,000 designs were calculated under the following conditions:

- Vane number ZN=23 (primary number);
- Blade number ZM=29 & 31 (primary numbers);
- Exit angle limited by 80 degrees;
- Maximum efficiency was the only target.

These additional conditions reduced the number of design variables and objectives, making it easier to find a solution.

As a result, the turbine's efficiency was improved up to 85.5% (see Figure 11). As one can see, the main gain in the turbine's efficiency was already achieved after 20-30,000 designs, while a very moderate, but clear improvement was obtained after that.

As already mentioned, 33 input variables were used in the design of the small axial-flow turbine. Obviously, not all these parameters equally affecting the turbine's performance. A sensitivity analysis was performed using the mF tool [4] to detect the most important input variables. This enabled the exclusion of certain variables

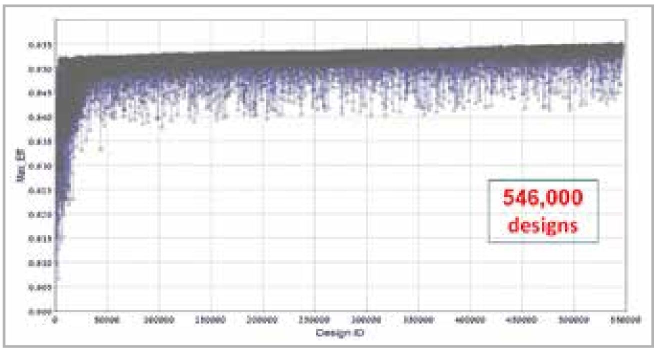


Fig. 11 - Efficiency-only optimization

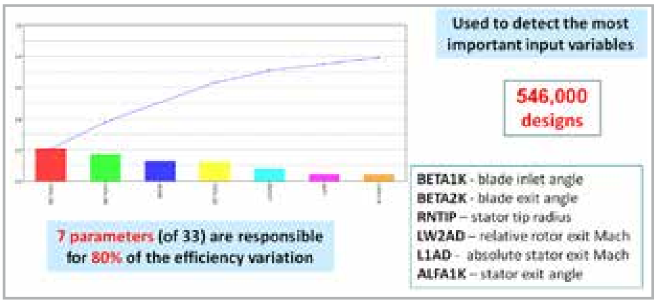


Fig. 12 - Sensitivity analysis

from the optimization, reducing the required computational effort. This analysis is particularly useful to better understand the physical model. In particular, sensitivity analysis functions can be used in the Response surface method (RSM) training process.

Figure 12 illustrates some of the sensitivity analysis results. The following 7 of the 33 parameters were responsible for 80% of the efficiency variation:

- BETA1K – blade inlet angle
- BETA2K – blade exit angle
- RNTIP – stator tip radius
- LW2AD – relative rotor exits Mach number
- L1AD – absolute stator exit Mach number
- ALFA1K – stator exit angle

The second section from the turbine disk, next to the hub, was found to be the most important section for maximum efficiency.

It is to be noted that the huge amount of data generated through the optimization process becomes difficult to treat: we found that the mF operations in the working table became slow at about 500,000 designs. Large tables cannot be treated by MS Excel due to its limit of 1,048,576 rows and 32,000 graph points. The optimization process was therefore split to cater for these limitations, and only the feasible designs were treated in post-processing analysis.

As a result of the optimization, the turbine exit angle was improved by 2÷6 degrees, while the efficiency was increased by 1.5%. One may question how meaningful this result is. Visual inspection of Figure 7 demonstrates a sufficient improvement in optimized efficiency compared to the manual design that was based on a few dozen attempts by a turbine design expert. However, some quantitative estimation of the performance improvement gained through the optimization is desirable. To this end, we used the classical Smith chart [5], which is a map that describes the empirical correlations between the efficiency of the state-of-the-art axial turbine stages, and the loading and flow coefficients, and which is widely accepted as feasible for use during preliminary turbine design. According to the Smith chart, the turbine efficiency achieved with the manual design was about 4% below the maximum efficiency of the axial turbine under consideration. This meant that the efficiency improvement of 1.5% achieved with design optimization had closed about 40% of the potential gain in turbine efficiency. Other measures, such as the advanced blade geometries, hub and tip contouring, abradable seals, stacking, and 3D blade design can close the remaining gap.

Conclusion

Fast computer code and the advanced modeFRONTIER optimization and analysis tools allowed us to perform an optimal mean-line design of a small axial turbine with more than 2,000,000 configurations. The time that was spent on this project, about 80 work hours, is usually sufficient to manually check a few dozen designs only. 33 parameters and 139 constrains were taken into account. The turbine exit angle was improved by 2÷6 degrees, and its efficiency was increased by 1.5%, closing about 40% of the potential efficiency gain for this type of turbine. We recommend using mF models with no more than 500,000 designs simultaneously.

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

References

[1] Harris, M.M., Jones, A.C. and Alexander, E.J., "Miniature Turbojet Development at Hamilton Sundstrand: the TJ-50, TJ-120 and TJ -30 Turbojets", AIAA paper 2003-6568, Sept. 2003.

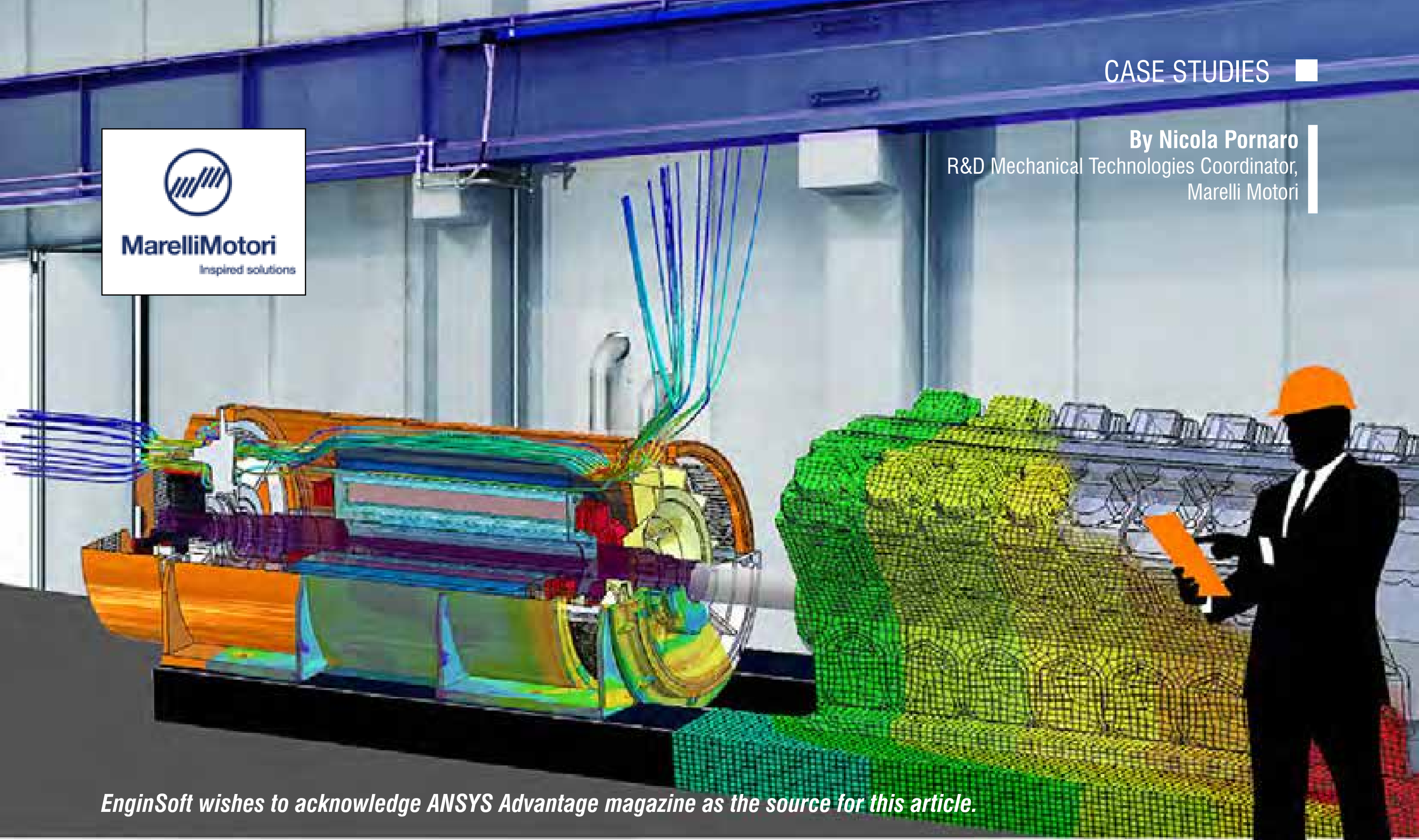
[2] Baturin, O.V., Popov, G.M., Kolmakova, D.A. and Novikova, Yu.D., "The best model for the calculation of profile losses in the axial turbine", IOP Conf. Series: Journal of Physics: Conf. Series 803 (2017) 012017.

[3] Venediktov, V.D., Granovsky, A.V., Karelin A.M., Kolesov, A.N. and Mukhtarov, M.X., "Atlas of Experimental Characteristics of Plane Cooled Gas Turbine Blade Cascades", Moscow, CIAM, 1990.

[4] modeFrontier User Manual

[5] Smith, S.F., 1965, "A Simple Correlation of Turbine Efficiency", Journal of Royal Aeronautical Society, Vol. 69, pp. 467-470.

By Nicola Pornaro
R&D Mechanical Technologies Coordinator,
Marelli Motori



EnginSoft wishes to acknowledge ANSYS Advantage magazine as the source for this article.

Electrifying Solutions for Motors and Generators

The market for electric power generation equipment is growing more competitive every day, with customers demanding more reliable, eco-friendly products at lower cost. Marelli Motori meets these demands using ANSYS Maxwell, ANSYS Mechanical and ANSYS CFD in multiphysics simulations to deliver the tailor-made solutions their customers have come to rely on. More recently, they have begun using ANSYS Discovery Live to obtain instantaneous simulation results with every on-the-fly change to a product's geometry or operating conditions, greatly reducing design time.

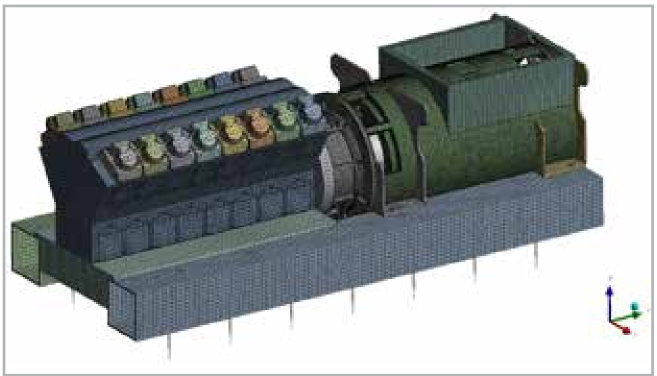
Electric motors and generators contain rotating magnetic coils through which electrons flow. The resistance of electrons flowing through wires, together with the friction generated by rotating devices, causes heat to build up. Energy lost as heat is unavailable to do work, reducing the efficiency of the motors and generators. Excess heat can also cause structural problems as temperature builds up in structural components and induces stress. Heat can be dissipated with cooling airflow, but the physics of the airflow must be optimized for maximum effect.

Because all these physical effects are happening simultaneously, a multiphysics simulation approach is needed. Marelli Motori engineers use ANSYS multiphysics solutions to custom-design motors and generators to solve challenges in hydropower, cogeneration, oil and gas, civil and commercial marine transport, military applications, and ATEX applications involving motors and generators in explosive atmospheres, among other applications. (ATEX consists of two EU directives describing what equipment and work space is allowed in an environment with an explosive atmosphere.)

Mechanical, Flow And Electromechanical Multiphysics Solutions

Marelli Motori engineers use ANSYS Mechanical to optimize the design of the frame, shields, cooling fan, motor shaft and generators. Structural simulations focus on reducing the weight of these components while optimizing stiffness. The R&D Team of Marelli Motori also simulates the response of the machine to the static and dynamic forces that are generated by the rotation of the rotor; excessive forces could lead to

> *"When simulating a heat exchanger on a closed alternator, an experienced ANSYS CFX user analyzed five different designs in eight hours; with ANSYS Discovery Live, the same engineer reached an optimal design in two hours."*



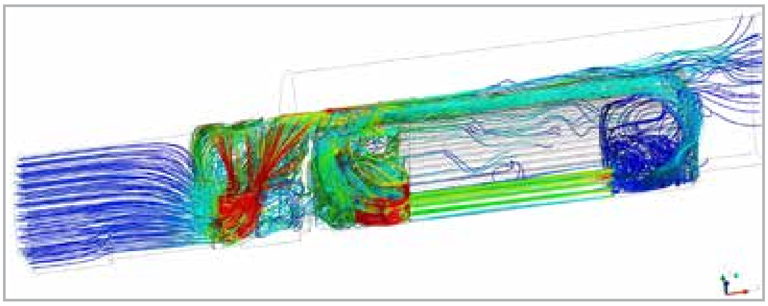
Example of a genset with Marelli Motori's alternator installed in a hospital in Germany

component failure through deformation, crack formation or fatigue. Using ANSYS Workbench as a common platform to perform multiphysics simulations, Marelli Motori engineers run ANSYS CFX simulations along with structural ones to determine the design that best combines optimal structural integrity, thermal efficiency and cost reduction. The rotor assembly (including single or double cooling fans, depending on the machine air circuit), the stator and the heat exchangers (when needed) are the core thermal exchange components of the motor or generator. ANSYS CFX computational fluid dynamics (CFD) simulations increase the cooling efficiency and thermal exchange with the surroundings by optimizing the airflow through the machines. This reduces hot spots inside the generators and motors to increase efficiency and maximize power output.

Finally, adding ANSYS Maxwell to Mechanical and CFX in multiphysics simulation completes the optimization process. The only way to reduce forces that create motor vibrations is to extract the magnetic forces using Maxwell and export them into a Mechanical analysis to evaluate the harmonic response of the frame. Maxwell is also used to identify hot spots in the coils and combine this analysis with a CFX calculation to locally optimize the design and improve the heat exchange. ANSYS Multiphysics simulations yield higher-quality results in 60–70 percent less time than other simulation products that Marelli Motori engineers have used in the past.

Manufacturing Challenges

Even after the design has been optimized using mechanical, flow and electromechanical simulations, the challenge of building the motor or generator most efficiently and effectively remains. Marelli Motori engineers want to facilitate the construction operations while keeping mechanical safety and reliability for each operating condition firmly in mind. This is the most challenging part of the engineering workflow, because while the engineers are trying to design a family of components to optimize heat extraction from the machine, they must simultaneously consider constraints regarding shape feasibility, production cost and ease of final assembly. Using ANSYS Mechanical and ANSYS CFX together in a multiphysics simulation guides the engineering team to the best manufacturing process. A recent project to develop a new series of small alternators with the latest technological improvements took much less time using ANSYS simulation.

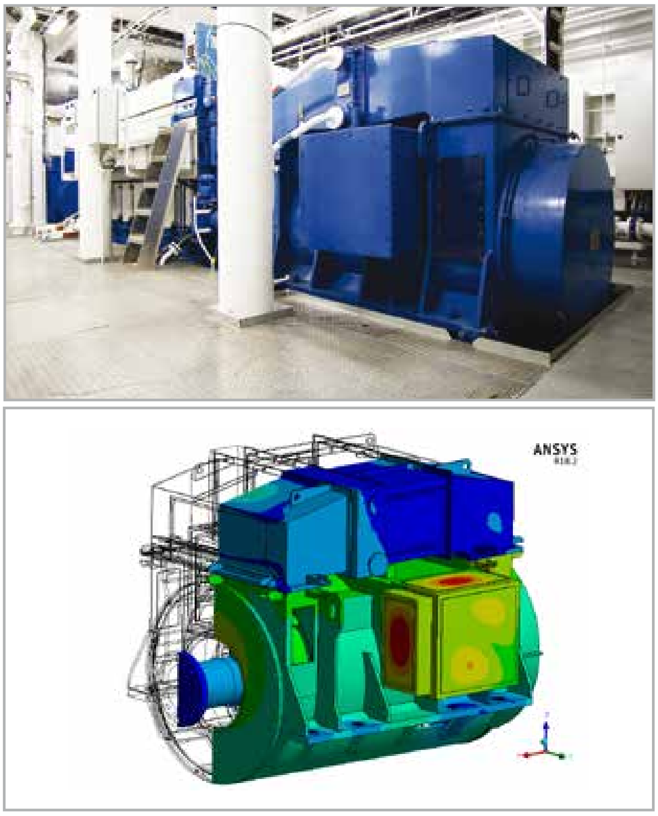


All the CFD simulations that lead to a redesign are subsequently evaluated in a test room. Here, some of Marelli Motori's motors for industrial applications are being tested.

Application Examples

Obviously, the importance of the various design parameters changes with each application. In marine applications, motors and alternators must be silent with very low vibrations to avoid ruining the experience of the ship's passengers. Structural finite element analysis and harmonic response calculations using ANSYS Mechanical must be performed on the frame and other components to reduce sound and vibrations.

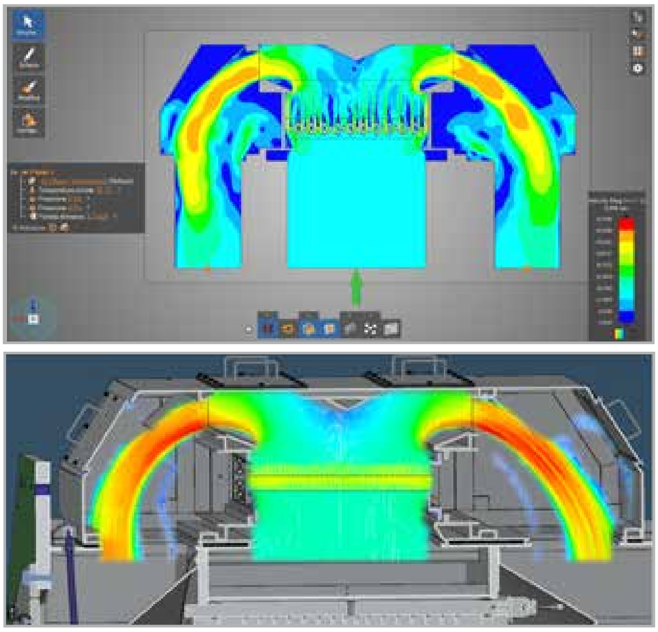
A genset is a combination of an internal combustion engine with an electric motor or alternator, used as a standby electric power supply. Vibrations from the diesel engine can excite natural frequencies and harmonic responses in the system. Marelli Motori engineers run modal analysis in ANSYS Mechanical to find these frequencies and harmonic responses, which vary according to operating conditions, to analyze the dynamic behavior of the alternator. This is followed by a collaboration between the customer and the genset designer to avoid any possible resonances of the entire genset with the surrounding structure for each design project. If this upfront analysis was not done, and the completed genset generated vibrations and structural noise inside a vessel,



Max power generator in a marine application and simulation of its alternator

correcting the problem would result in tremendous additional costs and project delays.

In power generation applications, increasing efficiency is the most essential step. This mainly involves applying CFD simulations to improve the airflow to cool the machines and coupling the results with EM simulations that optimize the electrical parts by reducing losses. Marelli Motori engineers perform this multiphysics simulation daily. All modifications introduced after numerical simulations are evaluated in a test room to demonstrate benefits in terms of temperatures and efficiency according to international norms.

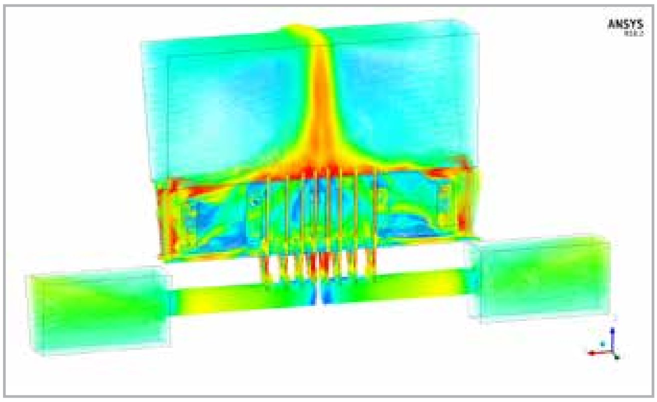

Example of a heat exchanger simulated using ANSYS CFX. An expert user completed five simulations in eight hours. Using ANSYS Discovery Live, a user completed many simulations in two hours to achieve an optimal design.

"The engineers used ANSYS Maxwell to identify hot spots in the coils and combined this analysis with an ANSYS CFX calculation to improve the heat exchange."

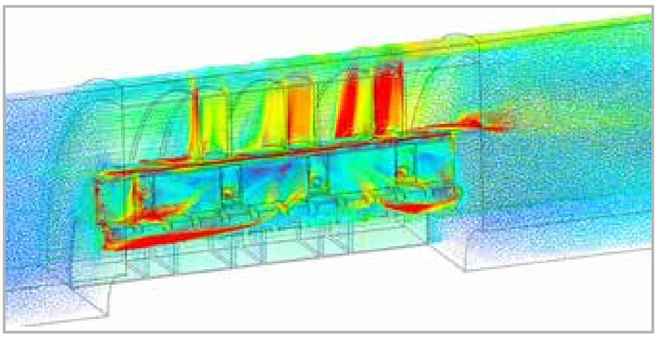
Using Simulation For Ideation

Marelli Motori was one of the first companies to adopt ANSYS Discovery Live when it was released early in 2018. Discovery Live is the first simulation solution to enable engineers and designers to make changes to geometry and other properties while a simulation is running and instantaneously view the results of these changes. With their commitment to promptly satisfy their customers with high-quality, reliable, customized products, Marelli Motori realized that such rapid simulation results would help them to react to their customer's needs faster. In one case involving simulation of a heat exchanger on a closed alternator, an experienced CFX user analyzed five different designs in eight hours; with Discovery Live, the same engineer reached an optimal design in two hours, a savings of six hours. ANSYS multiphysics simulations helped Marelli Motori engineers to design the best components for their customized electric motors and generators and become more competitive in the worldwide market. Their customers appreciate the increased efficiency, cost reduction and shorter development times, along with the greater reliability provided by the synergy between Marelli Motori and ANSYS simulation. ***www.marellimotori.com***

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



ANSYS CFX is used to optimize the cooling channels inside a rotor.


ANSYS CFX rotor simulation

Intermarine Shipyard tests Flownex SE for its naval piping systems

Software found to save time in the design phase

by Andrea Villa and Leonardo Carassale
Intermarine

There are various piping systems that convey many different fluids on board a vessel. Each fluid must reach its user at the right pressure and flow conditions. Accessories such as valves, bends, fittings and pipes induce pressure losses (as a result of factors such as pressure (p), flow rate (q) and pipe size (diameter, A)). The designer has to calculate these probable pressure losses in the pipeline in order to select (or verify) the size of the pump to be installed in the piping system to prevent a number of possible problems. Usually, these calculations to predict pressure losses are performed "manually" using the procedures described in the technical literature, such as the method of equivalent lengths, with the help of software such as Microsoft Excel or similar, and with the lengths and the fittings information being derived from one-line diagram (2D CAD software). The Shipyard wanted to test the capability of the 1-D computational fluid dynamics (CFD) software known as Flownex Simulation Environment (SE), provided by EnginSoft S.p.A., as a pipeline solver for its naval piping systems.

The results obtained with Flownex were compared with the "manual / classic" method described above and also with actual field data from on-board measurements.

CFD model

The testing activity focused on the "auxiliary seawater cooling system", in which we concentrated on different lines such as the suction and delivery lines of the pumps, and the inlet and outlet lines of the diesel generator. The Flownex model was designed with the following characteristics:

- ***Fluid*** - the fluid was customized to match the average seawater characteristics (such as density, kinematic viscosity, and temperature) Nodes and fittings - the pipelines were designed with their characteristic values using the software elements
- ***Pumps*** - these items were modeled by introducing the relevant performance curve (data provided by supplier) P=f(Q)

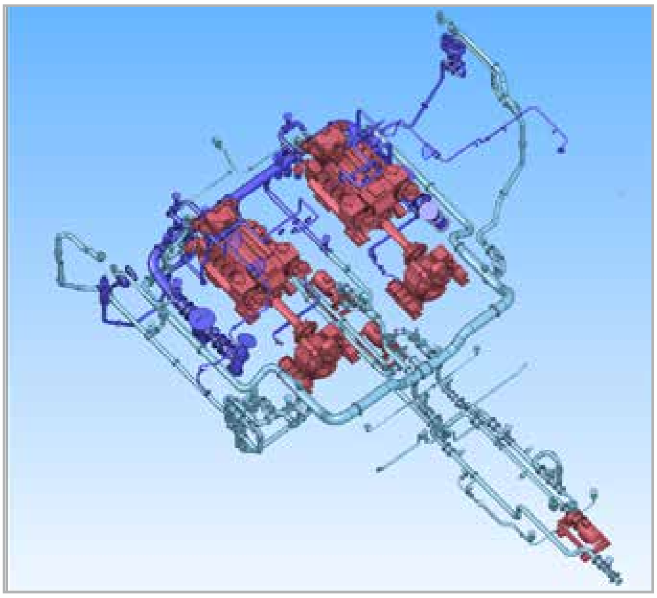


Fig. 1 - Excerpt of Auxiliary Piping System 3D Model (CADMATIC SW; pumps inlet and outlet)

Boundary conditions -> for each suction and delivery line, these were set with P (for the inlet) and Q (for the outlet).
Some special fittings, such as expansion joints that had no similar components already available in the Flownex library, were replaced with "basic pipe" elements of equivalent length that corresponded to their concentrated losses.

Numerical results

The pipelines were calculated at two different flow rates: the lower at approximately 39 m^3/h, corresponding to the operating point of the pump (for improved efficiency under design conditions), and the higher at approximately 112 m^3/h, corresponding to the flow rate when all the users are running the cooling system simultaneously (the worst potential condition, furthest from the design conditions).Lower flow rate case (39 m^3/h)
Notes:

- *) Boundary conditions may differ for intermediate or final lines due to a different result in previously calculated lines
- **) Negative values of pressure drops are related to a descending pipe, because of the gravitational effect.

The Flownex results showed a slight increase in pressure loss values (1) indicating a more conservative behavior in the software with

Pipeline	Manual / Excel P [bar]	Flownex P [bar]	Experimental Results P [bar]
Boundary Conditions	1,247	1,247	-
ΔS_{1-2} (39 m^3/h) Suction line losses	0,163	0,164	-
Pump differential (39 m^3/h)	3,4	3,4	-
ΔS_{3-4} (39 m^3/h) Delivery line losses	0,077	0,081	-
Node #81 Delivery Node pressure	**4,407**	**4,402**	**-**
Inlet and Outlet lines from the diesel generator (Fig. 3)			
Boundary Conditions	4,407*	4,416*	-
ΔS_{5-6} (45 m^3/h) Inlet line losses	0,807	0,838	-

Pipeline	Manual / Excel P [bar]	Flownex P [bar]	Experimental Results P [bar]
ΔS_{7-8} (10,8 m^3/h) Inlet line losses	0,129	0,282	-
ΔS_{9-10} (10,8 m^3/h) Inlet line losses	-0,095**	-0,074**	-
Genset concentrated losses	1,394	1,394	-
ΔS_{10-11} (45 m^3/h) Outlet line losses	-0,177**	0,473	-
Node #81 Outlet Node pressure	**2,349**	**1,504**	**-**

respect to the "manual" calculation. The same behavior is shown in (2), except for the Outlet line, because of the orifice device installed (these elements require further investigation, beyond the scope of this article).
Notes:

- Experimental results from on-board measurements

In this case (112 m^3/h), interesting also due to the availability of field data measured on the suction line, it can be noted that both calculations (Excel with 0,671 bar, Flownex with 0,684 bar of pressure drop) have a conservative behavior related to the pressure gauge measurements (0,397 bar), thus predicting a higher pressure loss than the ones measured directly on board. No actual data is available for the delivery line, so we could only compare the "manual" calculations with the Flownex ones. The first returned a pressure drop of 0,546 bar, the second reported a pressure drop of 0,602 bar. In addition,, in this case, the tested software displayed more conservative behavior.

Conclusions

In summary, the study led to the following conclusions:

- The aim of the study, in terms of calculating the pressure loss in the pipeline for design purposes, was achieved.
- The software provides results that are comparable to the traditional method, leading to the same results in terms of pipe dimensions or pump size selection.

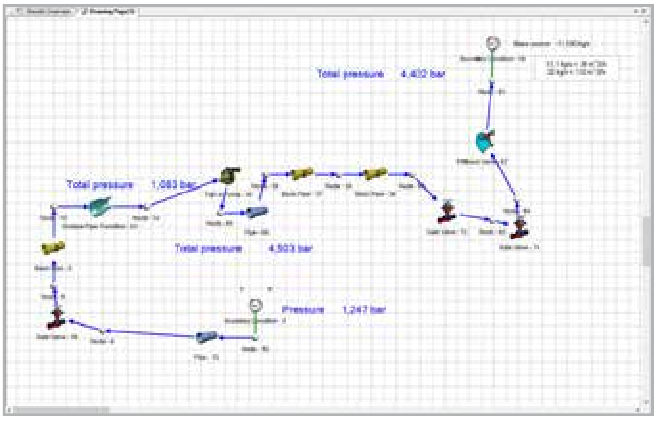


Fig. 2 - Suction and delivery lines of the pump (39 m3/h)

Pipeline	Manual / Excel P [bar]	Flownex P [bar]	Experimental Results P [bar]
Suction and Delivery lines of the pumps (Fig. 4)			
Boundary Conditions	1,247	1,247	-
ΔS_{1-2} (112 m^3/h) Suction line losses	0,671	0,684	0,397
Pump differential (112 m^3/h)	2,4	2,4	2,4
ΔS_{3-4} (112 m^3/h) Delivery line losses	0,546	0,602	-
Node #81 Outlet Node pressure	**2,430**	**2,506**	**-**

Higher flow rate case (112 m^3/h)

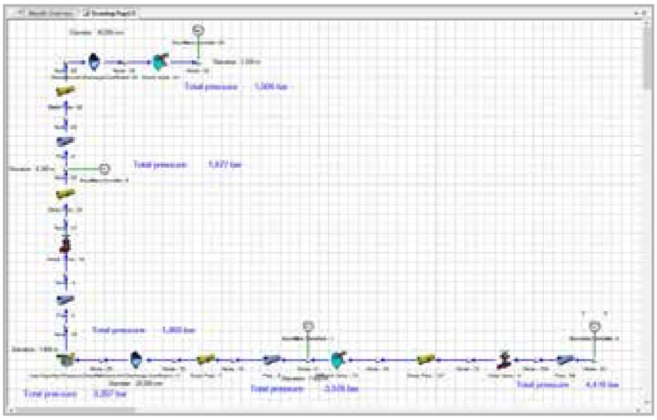


Fig. 3 - Inlet and outlet lines of the diesel generator

- The software can be a valid ally to save time during the design phase.
- Some fittings, such as restrictors (orifices) need to be better modeled to be successfully incorporated into the simulation.
- The comparison between the on-board measurements and Flownex could be extended to all systems in order to "calibrate/ ensure convergence" with the final software values.
- Further developments could include the use of Flownex SE software for all ship piping systems.

For more information
Erik Mazzoleni - EnginSoft
e.mazzoleni@enginsoft.com

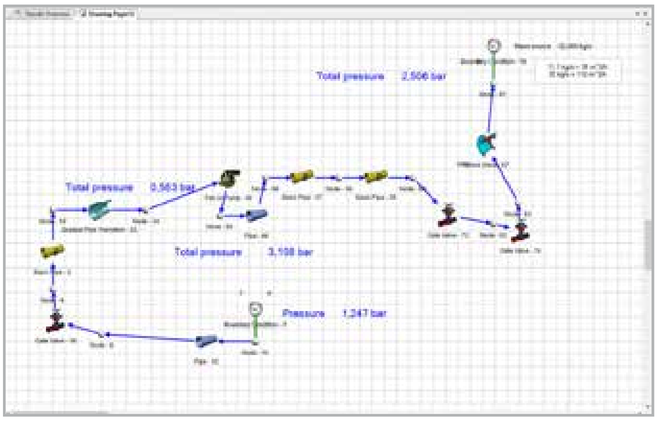


Fig. 4 - Suction and delivery lines of the pump (112 m3/h)

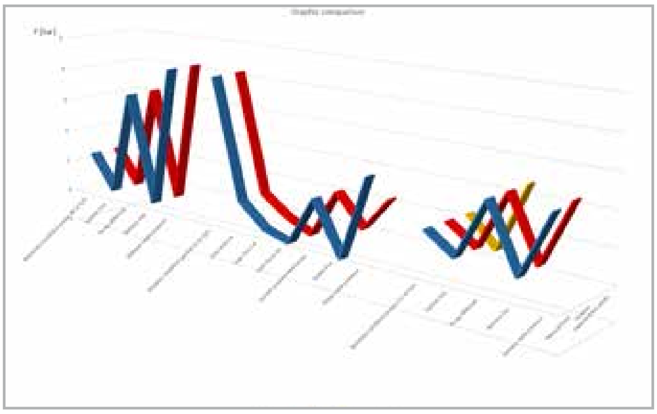
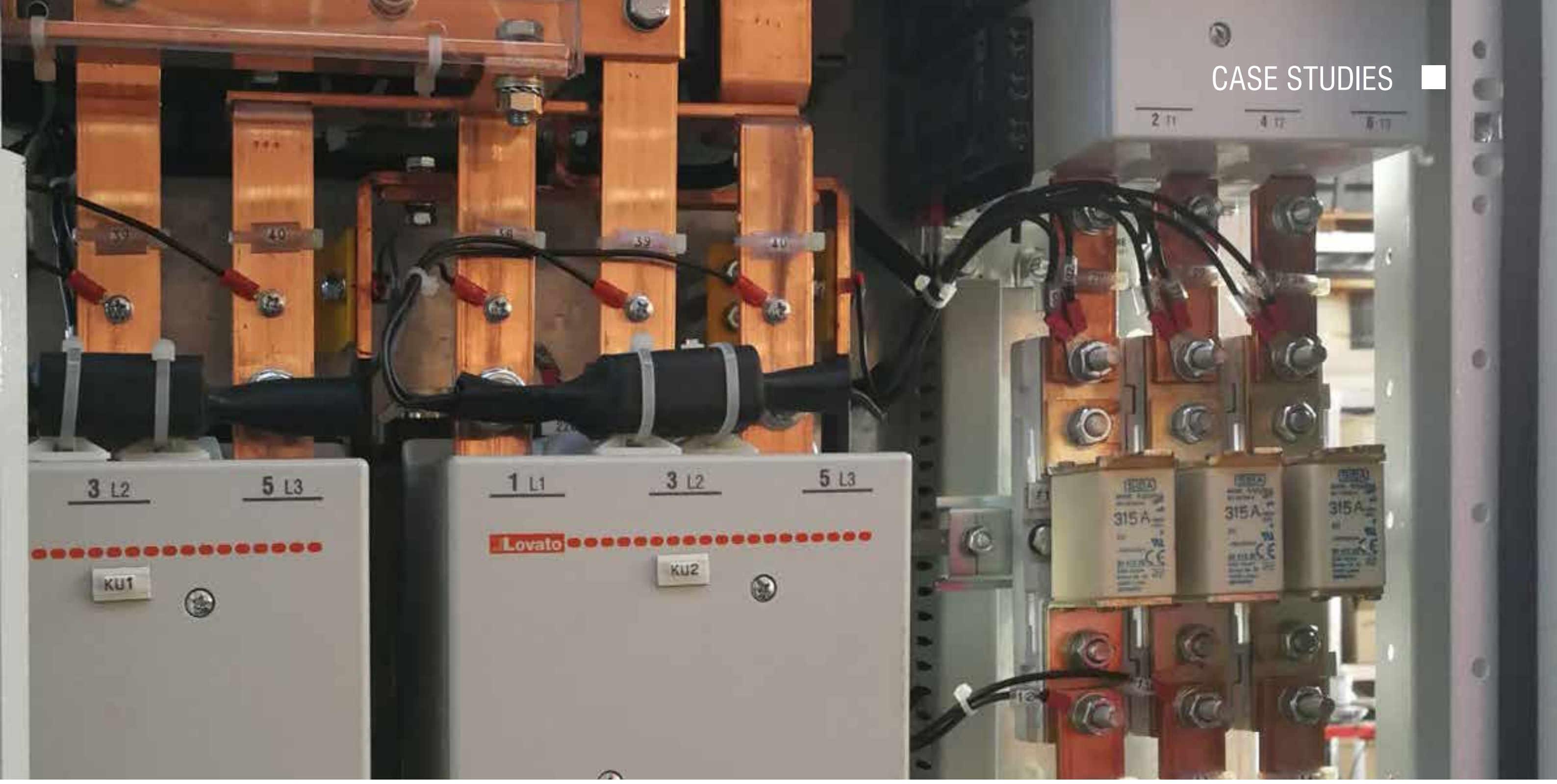


Fig. 5 - Graphic comparison of the pipelines



Using simulation and validation to build trust in new tools

A multibody simulation of a switch disconnector

By Iacopo Guaiatelli[1] and Gian Marco Colorio[2]
1. LOVATO Electric, 2. EnginSoft

Changing the way a company moves from an idea to a physical product is not always an easy task, especially as designers first have to develop trust in the results provided by any new tool. This article describes the multibody simulation of a switch disconnector and its validation with experimental tests, undertaken by LOVATO Electric for the purpose of conducting a detailed evaluation of a potential new tool to design the mechanical part of its electrical devices. This type of "field test" of potential new tools is also valuable for skills and knowledge transfer.

In order to reduce the time it takes to release a new product, companies are changing their design processes by increasing their use of CAE tools year on year. This is, for example, the strategy that LOVATO Electric is implementing: streamlining its design process through the use of simulation tools from the earliest stages of the design process.

Changing the way a company moves from an idea to a physical product is not always an easy task, especially as designers first have to develop trust in the results provided by any new tool. Before making the decision to adopt RecurDyn as the main tool to design the mechanical part of its electrical devices, LOVATO Electric went through a very detailed evaluation process consisting of the construction of a multibody dynamics (MBD) model of a switch disconnector and its validation by means of experimental tests.

Switch disconnectors

Switch disconnectors are electrical devices suitable for various applications such as electric equipment, machinery and power distribution, to perform quick-make, quick-break operations in low voltage circuits. These devices convert the potential elastic energy stored in the springs into kinetic energy for the mobile contacts in the poles. A quick-make, quick-break operation lasts about 5 milliseconds, so the velocities involved are extremely high.

An optimal design of the switch disconnector control is critical for high performance. In this type of mechanism, performance is identified as the time it takes to switch from ON to OFF during a break maneuver, as well as robustness and durability (over time, the arc generated during ON-OFF operations damages the copper parts).

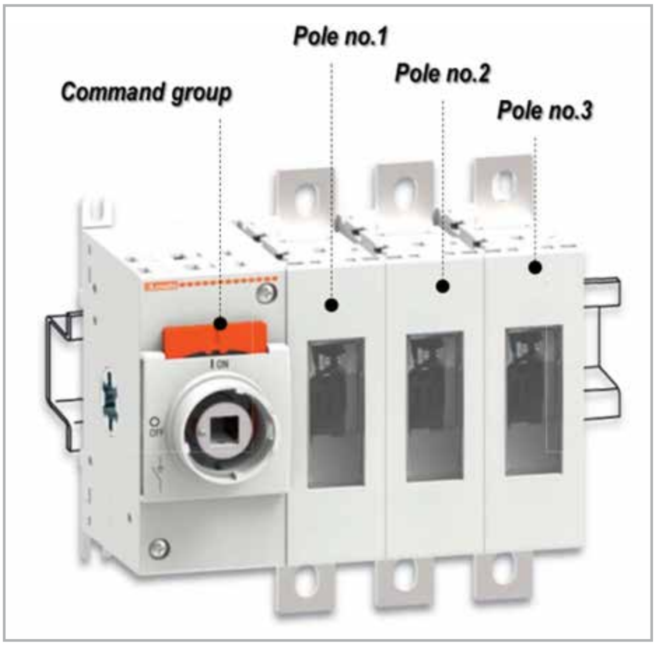


Fig. 1 - Switch disconnector architecture

After a period in which LOVATO Electric, shadowed by EnginSoft's multibody team, tested RecurDyn's capabilities, it was able to insert RecurDyn correctly into its product design cycle. LOVATO Electric is now able to predict the dynamics with high accuracy, and to identify the optimal configuration, increasing effectiveness and requiring the use of the laboratory for validation purposes only.

Why LOVATO Electric chose RecurDyn

To understand how LOVATO Electric has changed the way it designs its products, we need to review the original process. In the past, each new product was the result of a rigid multibody analysis, based on experience gained over the years by designers, and from both mechanical and electrical experimental tests. This method was very expensive in terms of time and resources and sometimes, too many iterations were needed to meet the specifications because of the use of inappropriate engineering tools.

Traditional embedded multibody CAD software lacks the three-dimensionality of the contact and cannot take into account the flexibility of the parts. These aspects make them unsuitable for applications where the contact surfaces are highly processed (such as in the switch control assembly), and the deformation of the body affects the delay with which the quick-break, quick-make operations are carried out.

LOVATO Electric was looking for a tool that could address all these aspects from the beginning of the design process, helping it to reduce the number of prototypes. That was when RecurDyn came into play with its unique features for detailed contact analysis (GeoSurface) and its proprietary technology for Multi Flexible Body Dynamics (FFlex).

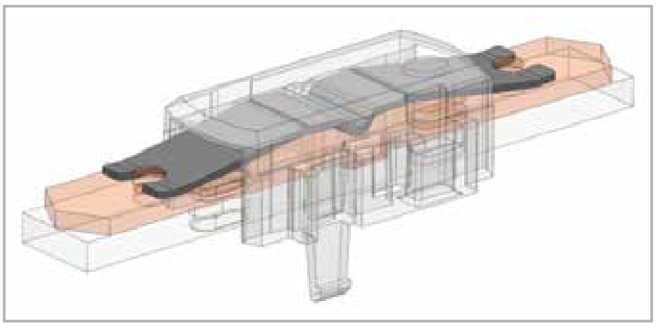


Fig. 2 - Leaf-springs in the pole mechanism

Why the FFlex method

The flexibility of certain bodies was fundamental in the switch disconnector that LOVATO Electric decided to analyze to test RecurDyn's capabilities. In particular, there were two leaf-springs under great deformation in the poles (see Fig. 2). Being subject to large deformations and being in contact with other bodies by means of sliding contacts, these springs show non-linear behavior.

With RecurDyn, LOVATO Electric's engineers were able to simulate the correct assembly procedure starting from the springs in undeformed shape and loading them to the assembled position by simulating the actual assembly procedure. After the validation of the leaf-spring model against experimental tests (Fig. 3), using the Extract feature, it was possible to create a subassembly of the pole with the leaf-springs already in their loaded configuration, ready to be implemented in the entire switch disconnector model in as many subassemblies as necessary.

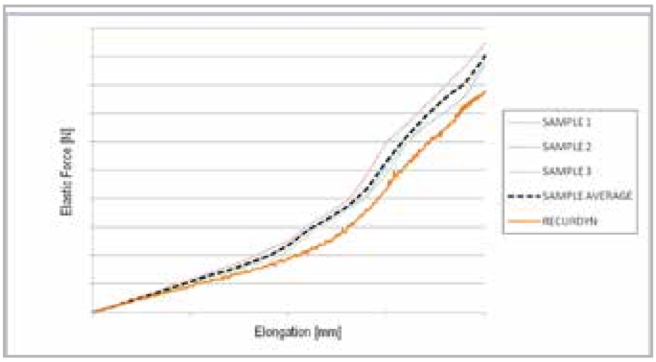


Fig. 3 - Validation of leaf-springs subsystem

The FFlex method allows the stress within the components included in the model to be evaluated (even with non-linear material properties), while considering the effects of the deformation of the body. This aspect is crucial to identify structural problems from the early stages of the design.

Why GeoSurface contact

The switch control assembly is the device required to translate a slow rotation of the handle to a rapid switching of the switch.

At the center of the control unit is a sophisticated cam system involving a central shaft, called the primary shaft, and two other shafts at right angles to it, called secondary shafts.

These three shafts communicate via cam-type surfaces whose shape plays a very important role in performance. This part of the mechanism is impossible to model with a standard joint and it is here that GeoSurface contact was widely used, as it is able to efficiently manage the contacts

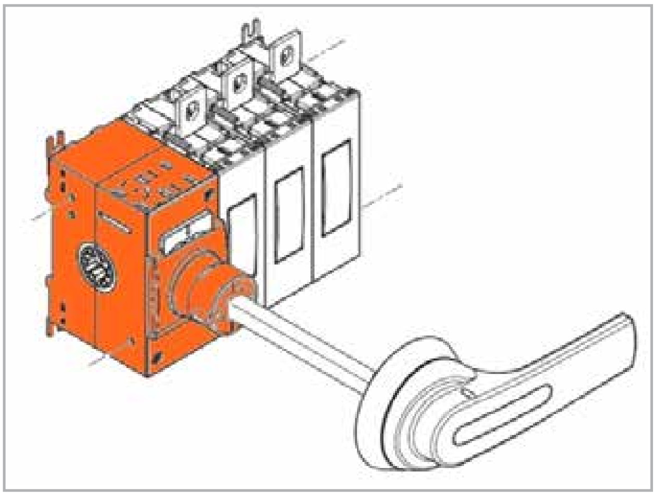


Fig. 4 - Switch control assembly

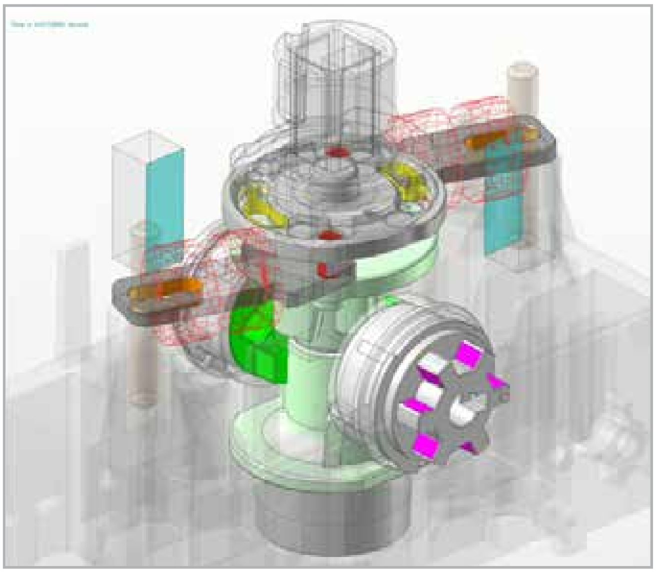


Fig. 5 - GeoSurface contacts within the switch control assembly

between complex surfaces characterized by extensive sliding between them.

Future developments and conclusions

As a next step in this project, LOVATO Electric wishes to remove most of the joints and use contacts instead, in order to take into account the effects of all the clearances. The company also wants to implement RecurDyn and the techniques learned during this study in the design of the dimensions of the next switch disconnector and other products. As a result of this project, in fact, LOVATO Electric has acquired most of the skills necessary to correctly simulate the switch disconnectors using RecurDyn and is now able to use it for its entire range of electromechanical products.

Visit also www.lovatoelectric.com

For more information
Fabiano Maggio - EnginSoft
f.maggio@enginsoft.com

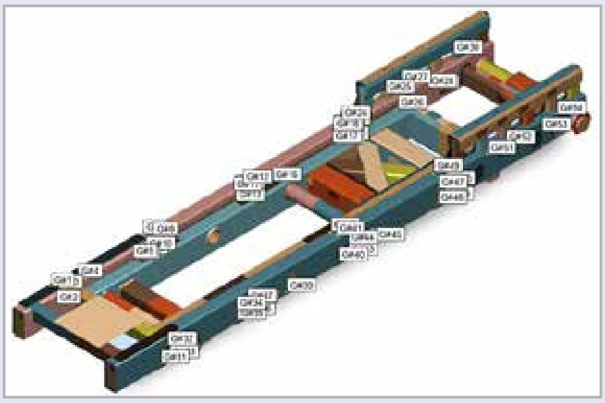
Review: Less pain to predict strain

Warehouse fire-fighting event

By Tim Hunter
Wolf Star Technologies, LLC

This review is based on original article that was first published in the ANSYS Advantage Magazine, Vol XIII, Issue 1, 2019, which can be accessed at: www.ansys.com/about-ansys/advantage-magazine/volume-xiii-issue-1-2019/less-pain-to-predict-strain.

An interesting article was recently released in the ANSYS Advantage Magazine showing the application of a novel technology to understand complex time domain loading of a Cummins generator skid. The article, written by Cummins Sr. Mechanical Engineer, Nathan Marks, details the steps involved in back calculating the operating loads using a tool developed by Wolf Star Technologies called True-Load™. In the article, Marks explains how the traditional techniques of using accelerometers does not provide the numerical fidelity and precision needed to accurately predict fatigue damage on the skid structure.



Recommended locations for 54 strain gauges from True-Load/Pre-Test based on the modal analysis data

The article demonstrates how the strategic placement of strain gauges using the True-Load™ software provides a robust mathematical model that is used to reproduce the operating loads. The dominant mode shapes in the structure are also identified with this technique. The baseline structure did not experience any durability issues. However, the design team needed to create some design changes to the structure. Marks demonstrates how he was able to use the operating loads from True-Load™ to help guide the refinement of the necessary design changes. He concludes that the technique can be applied to most structural issues on frames and most other systems on the generator.

Marks expertly showed how to adopt a new technology to an old problem at Cummins. He stated, "The Cummins engineers found this to be a huge value-add through both reduced development and testing time and overall reduced part cost by eliminating overdesign." This statement basically says it all. True-Load™ provides a solution that results in less pain to predict strain.

Testing the potential to offer CAE over the cloud in the SaaS paradigm

FSI optimization of industrial airplanes: the P180 Avanti EVO study

This technical article describes a fluid-structure interaction (FSI) optimization carried out for the P180 Avanti EVO vehicle, a business aircraft designed and manufactured by Piaggio Aerospace. It describes the use of an additional module for the RBF4AERO platform, rbf4aerpFSI, designed and validated to work with both commercial and open source solvers, to conduct the CAE analyses of a set of modifications to the P180's winglet shape. RBF4AERO is the software platform resulting from the EU-funded RBF4AERO project (FP7/2007-2013) to address the highly demanding CAE requirements of aircraft design and optimization to improve aircraft and component performance. The rbf4aeroFSI solver allows designers to perform single- and multi-objective optimization (SOO and MOO) using evolutionary algorithms (EA) assisted by metamodels, while also taking into account the elasticity of the deformable components of interest under steady state conditions using two computation fluid dynamics (CFD) methods: two-way and mode-superposition.

By Emiliano Costa
Rina Consulting

This article describes a fluid-structure interaction (FSI) optimization carried out for the P180 Avanti EVO vehicle, designed and manufactured by Piaggio Aerospace. The study was performed within the framework of Experiment n. 906 of the Fortissimo project, funded by the European Union's Horizon 2020 research and innovation program under grant agreement No. 680481. In particular, it numerically investigated, by means of the mode superposition method, the effect of a set of modifications to the winglet shape, which were applied with a mesh morphing technique based on radial basis functions. The computational fluid dynamics (CFD) analyses were carried out with both commercial (CFD++, ANSYS Fluent) and open-source (SU2) solvers, using the RBF4AERO suite's cross-platform FSI solver.

Background

The RBF4AERO project, which was funded by the EU's Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 605396, aimed to create software (referred to as the RBF4AERO platform) to handle the highly demanding requirements of aircraft design and optimization to considerably shorten the time needed to finalize the computer-aided engineering (CAE) analyses for improving the performance of an aircraft and its components.

Upon completion of the RBF4AERO project, some of the partners, with the support of the Piaggio Aerospace company, agreed to undertake some further exploitation activities.

One of these activities was a submission to participate in the Fortissimo 2 project, being funded by the EU Horizon 2020 research and innovation program under grant agreement n° 680481, with the main objective of offering CAE services using the RBF4AERO platform through the cloud-based Fortissimo Marketplace.

The platform's key elements are tools that existed prior to the RBF4ERO project that were improved and suitably integrated into a single comprehensive working environment over the duration of the project. Specifically, they include the morpher tool (MT) - the commercial standalone version of the RBF Morph technology; the optimization Manager (OM) - employed to run single- and multi-objective optimization (SOO and MOO) problems using evolutionary algorithms (EA) assisted by off-line trained surrogate evaluation models (metamodels); and the in-house developed adjoint solver for the OpenFOAM suite.

The basic function of the platform is to create the parametric of the CAE model through a meshless morphing technique based on radial basis functions (RBF) to enable the computational studies for which the RBF4AERO platform was conceived, which are:

- EA-based optimizations, including constraint SOO or MOO which can be coupled with the FSI option;
- Icing studies;
- Adjoint-morphing coupling optimizations.

The results obtained with the EA-based operational scenarios can be reviewed and post-processed using an embedded post-processing module. Furthermore, the platform can schedule and monitor simulation jobs and supports multi-user and multi-hardware management.

With particular reference to FSI studies, a solver called rbf4aeroFSI was designed, validated and implemented to work with both commercial and open-source solvers. The rbf4aeroFSI solver allows designers to perform SOO or MOO using EA assisted by metamodels, while also taking into account the elasticity of the deformable components of interest under steady state conditions, according to two methods: two-way and mode-superposition. It is worth specifying that the two-way method is based on the exchange of data between the CFD (loads) and the computational structural mechanics (CSM) (displacements) models, whilst the mode-superposition method is based on the importation of the natural modes and frequencies of the deformable parts, with the calculation of the modal forces and the actual displacement being performed directly in the CFD model.

Main objectives of Experiment 906

In Fortissimo Experiment n. 906, known as "Cross-Solver Cloud-based Tool for Aeronautical FSI Applications", the proposed aeronautical application consisted of the aero-elastic optimization of the winglet of the P180 Avanti EVO (see Fig. 1), a business aircraft designed and manufactured by Piaggio Aerospace.

The FSI optimization

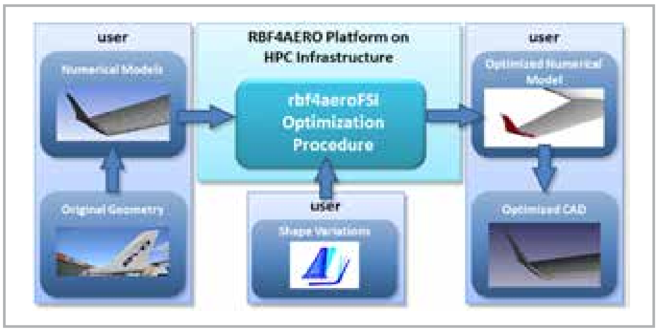


Fig. 1 - Workflow of the proposed FSI optimization

Shape modifications

The three shape modifications executed are twist, cant and sweep angles. In particular, the twist angle variation concerns the rotation of the winglet tip around the trailing edge while maintaining a fixed root, the cant angle shape modification changes the winglet angle with respect to the wing, whilst the sweep angle geometrical variation is achieved by translating the winglet position along the wing chord.

The RBF set-up controlling the surface mesh is shown in Fig. 2, where the red area on the left represent the moving points, while the green areas do not change their position during morphing. Fig. 2 (right) shows the final position of the source points, amplified 10 times to ease comprehension. The portion of the winglet between the moving and the fixed points is left free to deform by morphing.

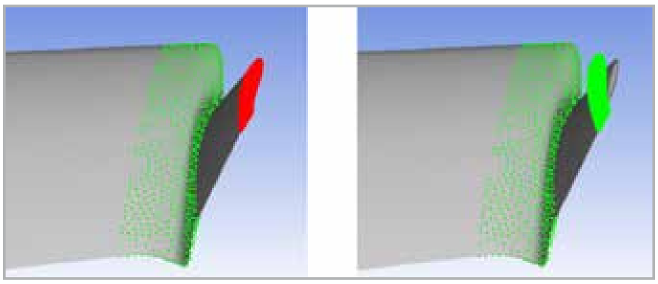


Fig. 2 - Winglet twist angle RBF set-up for surface mesh morphing

The RBF set-up that manages the volume mesh provides for the definition of a domain in which the morphing action is delimited. Due to degradation of cell quality from the morphing, combinations of the created shape modifications were verified by evaluating the quality of the resulting mesh in the range of variations of the selected angles.

Mesh and CFD model set-up and results for the baseline case

An unstructured hybrid computational grid with about 21 million cells was generated according to the specifications and settings typically used by Piaggio Aerospace to create a medium precision grid. In particular, a surface mesh formed by triangular elements was generated only from the imported CAD model. Then a series of layers was inflated from the surfaces to properly resolve the boundary layer to finally generate the tetrahedral cells in the remaining portion of the simulation volume. The transonic cruising conditions commonly used by Piaggio Aerospace were selected to perform the CFD simulations included in the FSI optimization. In particular, the flow regime enabled the identification of the main settings to be assigned in the CFD set-up, such as the angle of attack, the Mach number and the altitude conditions, while the influence of the engine was simulated through the correct boundary conditions being applied to the inlet and outlet.

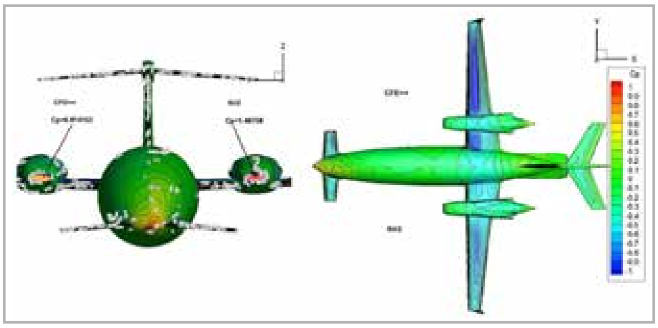


Fig. 3 - CFD results obtained for the baseline configuration of the P180 model

Fig. 3 from the front (left) and bottom (right) view respectively, depicts the comparison between the distribution of the pressure coefficient (Cp) over the surfaces of the aircraft obtained with SU2 and CFD++ in stable cruising conditions.

The fully developed solution generated for the baseline configuration was used to initialize the calculation of each design point (a baseline shape variant) of the FSI optimization, thus saving computing time.

CSM model and model analysis results

The CSM model includes the central parts of the aircraft. This model and its position and space with respect to the CAD model is shown in Fig. 4. While the winglets are modelled in detail down to the composite material used, the rest of the model is simplified by using plates and beams for the wing, and rigid elements for the engines, and by using spring elements to connect the wing and fuselage, maintaining the two extremes of the fuselage fixed.

Since the surfaces of the wet CFD and CSM did not match, it was impossible to achieve an FSI optimization using the standard two-way method. However, by exploiting the meshless property of the morpher tool, it was possible to interpolate the structural displacements to propagate them on the CFD mesh with a good level of accuracy. For this reason, the modal superposition method, which provides for the incorporating of structural modes in the CFD model, was chosen to implement the FSI analysis. According to this method, the FSI is solved as a reduced order method in which the structural behaviour of the system is condensed using a chosen number of modes, also called retained modes, each of which allows a single degree of freedom problem to be solved directly in the CFD solver.

Fig. 4 - The CSM model and its position with respect to the CAD model

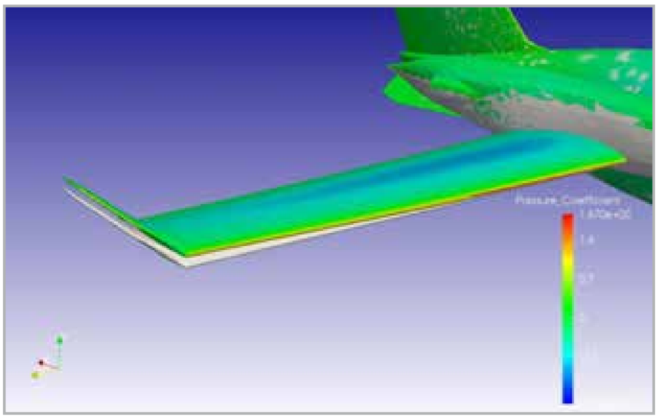


Fig. 5 - Comparison between the baseline (bottom-grey) and the deformed configuration

The first 30 modal shapes in particular were extracted, but most of them were discarded as being relative to the local vibrational modes of the plates. The modal shapes extracted from the CSM solver were used to generate a shape parameter for each one, adopting a similar strategy for the set-up of the RBF solutions to the one followed for the shape modifications previously detailed.

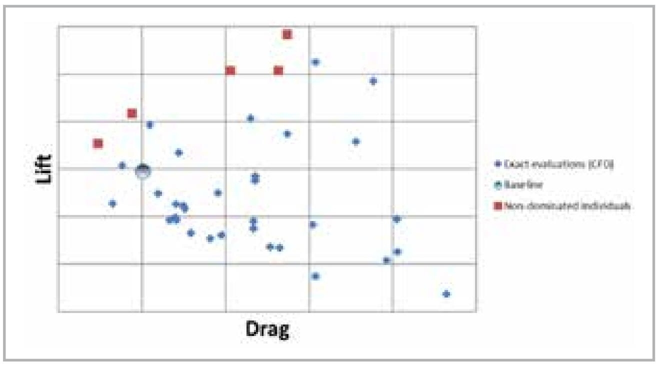


Fig. 6 - Main optimization outputs. Front of the non-dominated solutions together with all the candidate solutions evaluated during the optimization.

Optimization results

In relation to the optimization analysis, a two-objective optimization was run by selecting the drag function as the objective to minimize and the lift function as the objective to maximize in exploring the three shape modifications between +/-15°. As for the optimization settings, the full factorial method was selected to apply to the 27 design points identified from the exact CFD-based evaluations resulting from the design of experiment (DoE). Other settings concerned setting the child population size to 60 and the parent population size to 20, while the elite size was set to 5, the maximum number of exact evaluations was set to 40 and the maximum approximations was set to 500. Fig. 5 shows the comparison between the baseline (bottom-grey) and the deformed configuration (colored-top) of the wing for the baseline configuration. Fig. 6 shows the optimization's most important output, namely the distribution of the exact evaluations which highlight both the baseline configuration and the non-dominated individual configurations in different colors (the data set is confidential). As can be seen, the FSI optimization showed that there is room to effectively optimize the winglet in stable cruise conditions. Since non-dominated individual configurations can be generated using known combinations of shape modifications, one of these can be then identified as the optimal configuration to improve the aircraft's cruising performance.

Conclusions

Using the case of the winglet of a mid-sized business aircraft in cruise conditions, the RBF4AERO platform's effectiveness to perform a highly automated optimization study, while also accounting for the effect of the wing's elasticity, was demonstrated. This optimization was undertaken by adopting the mode superposition method, even though the wet surfaces of the CFD and FEM models did not match. This study showed the margins for improving the aerodynamic efficiency of an industrial vehicle, since the mesh morphing methodology can help to optimize the shape of the winglet in order to achieve, for instance, a reduction in fuel consumption during flight. The RBF4AERO platform confirmed itself to be a potential numerical means of offering CAE services over cloud infrastructures such as the Fortissimo Marketplace in the software as a service (SaaS) paradigm.

Acknowledgments

The consortium members (RINA Consulting, the University of Rome Tor Vergata's Department of Enterprise Engineering, the National Technical University of Athens, CINECA in Italy, the Italian National Research Council (CNR), and Piaggio Aerospace in Italy) wish to thank the European Union for the financial support provided. The Fortissimo project received funding from the European Union's Horizon 2020 research and innovation program under grant agreement No. 680481.

The paper full version and related references are available in the Proceeding of the International CAE Conference 2019 at http://proceedings2018.caeconference.com/sessions.html#aero

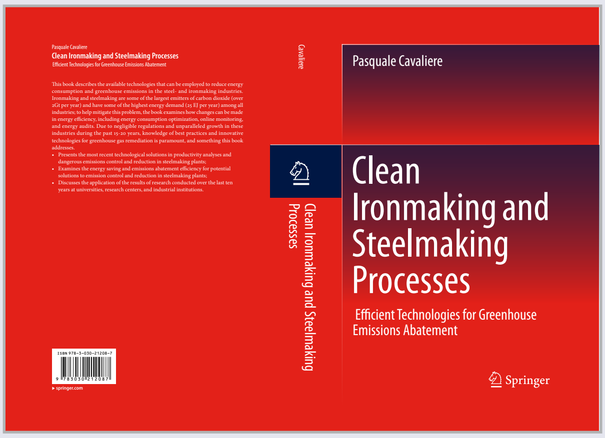
For more information
Emiliano Costa - Rina Consulting
emiliano.costa@rina.org

New book on clean ironmaking now available for pre-order

Provides best practices and innovative technologies for greenhouse gas remediation

A new book entitled "Clean Ironmaking and Steelmaking Processes - Efficient Technologies for Greenhouse Emissions Abatement" by Pasquale Cavaliere, professor of metallurgy at the University of Salento in Italy, is now available for pre-order from Springer International publishing.

This book describes the available technologies that can be employed to reduce energy consumption and greenhouse emissions in the steel- and ironmaking industries. These



producers are some of the largest emitters of carbon dioxide (over 2Gt per year) and have some of the highest energy demand (25 EJ per year) among all industries. To help mitigate this problem, the book examines how changes can be made in energy efficiency, including energy consumption optimization, online monitoring, and energy audits.

The author presents the most recent technological solutions in productivity analyses and dangerous emissions control and reduction in steelmaking plants. He examines energy saving and emissions abatement efficiency for potential solutions and he discusses the application of the results of research conducted over the last ten years at universities, research centers, and industrial institutions.

"The unparalleled growth of these industries coupled with the negligible regulations governing them has made the best practices and innovative technologies for greenhouse gas remediation vitally important, which is what I have provided in this book," states Cavaliere.

To pre-order the book, visit www.springer.com/gp/book/9783030212087.

Two-step approach to numerical simulation of fire and smoke propagation

Early stage use in design aids fire protection assessment of buildings and evacuation routes faster and more cost effectively

by Sandro Gori[1] and Chiara Crosti[2]
1. EnginSoft - 2. Freelance

Civil engineers make use of numerical simulations of fire and smoke propagation in the early stages of building design to assess a building's structural behavior, the fire resistance of evacuation routes and the speed and extent of the spread of a fire and smoke, among other things. This article explains how the simulation of a severe fire in a warehouse that had caused substantial damage was undertaken. It explores the use of the fire dynamics simulator (FDS) code, developed by the US National Institute of Standards and Technology (NIST), with a two-step approach to the analysis of the cause and spread of the fire that reduces the requirement for computational resources and the time required to execute the simulation.

Numerical simulations of fire and smoke propagation are widely used during the early stages of the design process to assess, among other things, the structural behavior of buildings, the fire resistance properties of evacuation routes and more.
For this project, the simulations were developed to represent the initial phase and the spread of a fire in a warehouse and included all the intermediate bulk containers (IBC) loaded with liquids that were stored in the building and which were partially involved in the fire.

The fire that was simulated had been severe, causing substantial damage to the building. It was predominantly the stocked materials that were involved in the fire. Most of them, in fact, were fully loaded IBC that had been piled up by type. The fire was confined to a limited area of the facility thanks to the intervention of the firefighters, who were able to completely extinguish the fire several hours after the first flames spread.

The interested warehouse consisted of three buildings, one of which, whose side walls were partially open, was involved in the fire. The gross area involved in the fire measured approximately 60m in length and 28m in depth, with a wall height of 4m. The ceilings of the warehouse consisted of barrel vaults made of concrete with bricks and steel rods and there were large openings between the sidewalls and the ceiling itself (see Fig. 1). This type of construction guarantees the supply of oxygen to a fire, as reported below.

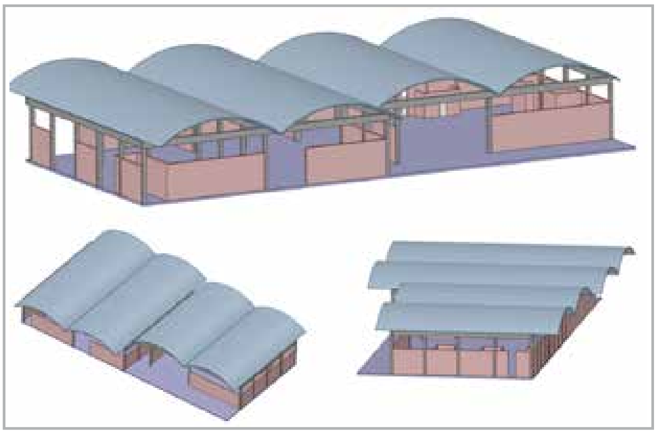


Fig. 1 - 3D geometry of the building

The IBC stocked in the warehouse contained several liquid materials, not all of which were inflammable and not all of which were directly involved in the fire. However, there were other combustible objects present in the building, like stacks of wooden pallets. Overall, the following materials were involved in fueling the fire:

- polystyrene insulating panels;
- wood (pallets);
- high-density polystyrene (external walls of the IBC);

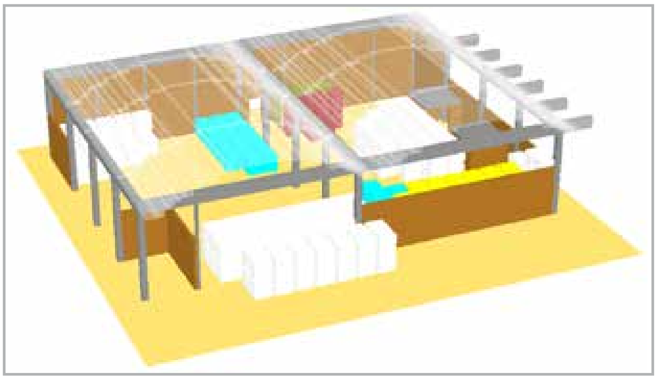


Fig. 2 - Geometric representation in FDS

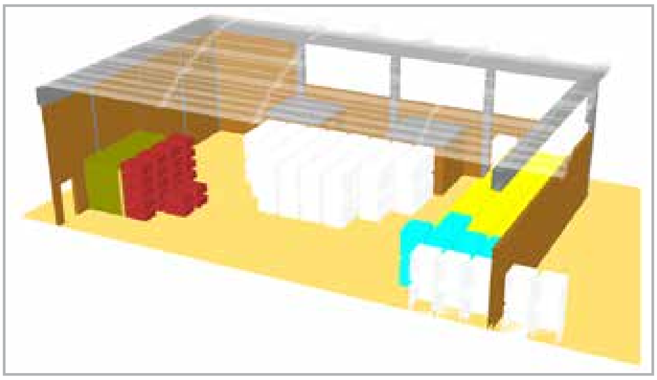


Fig. 3 - Detailed view of the model - different colors depict different materials

- vegetable oils (in the IBC);
- glycerin (in the IBC).

The main hypothesis regarding the first stage of the fire was that it had started as a result of some electrical issue that had occurred in the electrical power grid close to the warehouse. Specifically, the starting point of the fire was identified as having been an electrical fault in one of the main power control panels of the building. Thanks to the proximity of the polystyrene insulating panels and of the wooden pallets, the fire spread from this source to the nearby IBC, rapidly growing in size and eventually causing the ceiling to collapse.

In order to evaluate the hypothesis of the first stage of the fire (its cause and spread), numerical simulations were carried out using the Fire Dynamics Simulator (FDS) code developed by the US National Institute of Standards and Technology (NIST). This computational fluid dynamics (CFD) code is a large-eddy

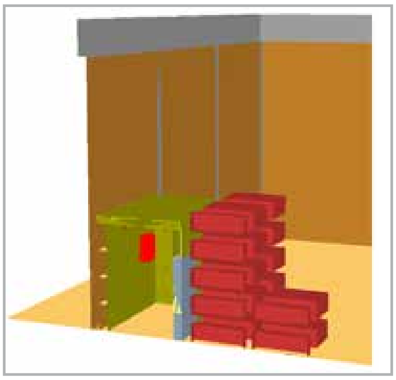


Fig. 4 - The burner (red box)

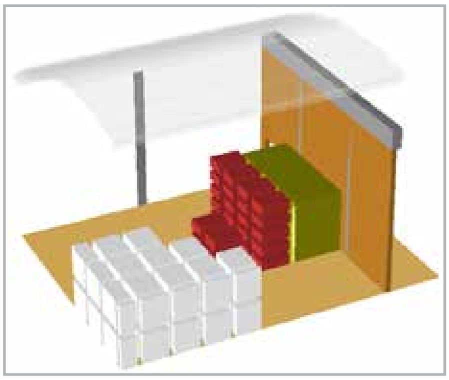


Fig. 5 - Reduced model

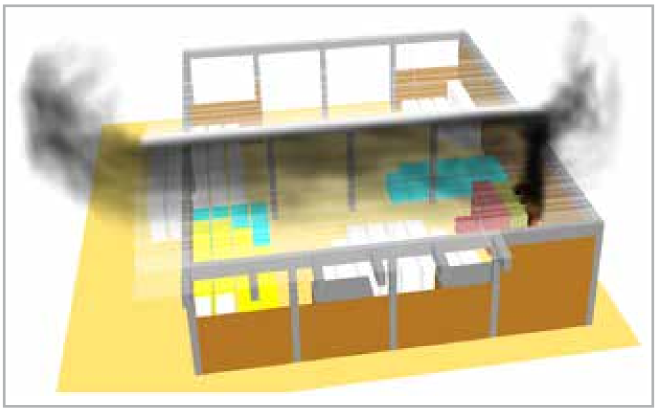


Fig. 6 - Fire and smoke 762s after the ignition

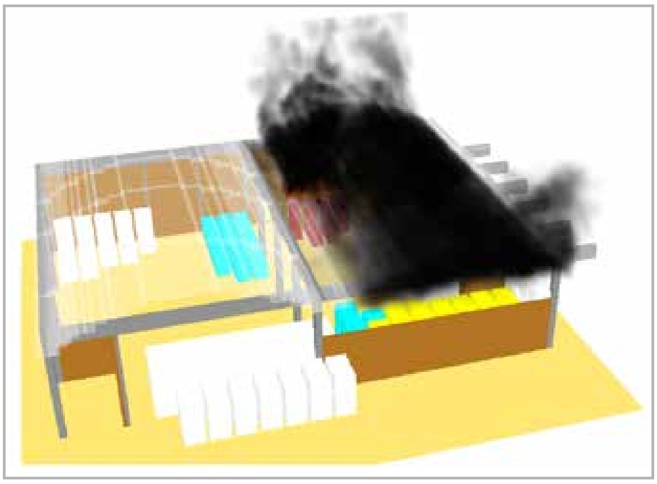


Fig. 7 - Fire and smoke after 1500s

simulation (LES) code for low-speed flows, with an emphasis on smoke and heat transport from fires, and is widely recognized as the standard tool for these types of applications as a result of its thoroughly verified and validated development process.

The computational domain of the model, which was discretized with a Cartesian mesh, was limited to a specific portion of the warehouse, which measured 33.6 x 31 x 15 m, to speed up the simulation. In the full model, this domain was divided into 24 meshes, each one consisting of 92,000 cells (over 2.2 million cells globally). The 3D geometry was able to be properly reproduced in the numerical model as a result of the chosen size of the cells (cubic cells with a 0.2 m edge size). The model included all the elements, both structures and fire-related materials/objects, that were required to accurately reproduce the building (see Fig. 2 and 3).
A specific object was used to represent the ignition source (the so-called burner) in the model. It was positioned in a small room, to emulate the same spot that the power panel was fitted in the actual warehouse (see Fig. 4). The burner's properties were represented in the model as a box emitting a net heat flux. These values were calculated from the nature of the ignition (an electrical fault) and from the analyses of the video recorded by the CCTV system.

To simplify the model and reduce the computational cost of the analyses, the behavior of the materials listed previously was imposed on the basis of - the so-called "ignition temperature" approach. This method makes it possible to avoid defining a pyrolysis model for each material and, conservatively, to approximate the involvement of the objects according to a specific Heat Release Rate curve for each material. When an object made of a specific material reaches the ignition temperature, it starts to release heat according to the curve (power over time) defined in the input file of the model.

There were two different steps to the numerical investigation strategy. Firstly, the analyses were related to the calibration of the cartesian mesh size and to the correct representation of the burner. A set of analyses was executed on a reduced portion of the domain that only included the burner and the room covered with the polystyrene insulating panels. Secondly, the reduced model was nested in the global one, to execute the full domain analyses and to post process the whole model.

This approach allowed us to obtain some extremely interesting results: in terms of the representation of the early stage of the fire, the reduced model was adequate and enabled the use of a mesh size that reduced the computation time for the analysis and the requirement for very large high performance computing (HPC)

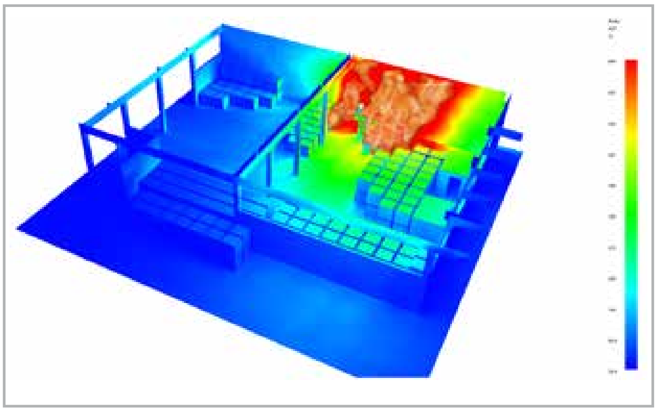


Fig. 9 - Spread of the fire and the adiabatic surface temperature of the object after 2634s

devices. The full model was run on 24 cores of a cluster and took 48 hours of computation time to evaluate two hours of the spread of the fire and smoke.

The global model represented the actual development of the fire and provided very significant results on the lead-up to the moment that the structural collapse occurred. We were also able to define the behavior of the materials and of the ignition source of the fire as they corresponded to the video recording and the data collected after the event (see Figs. 6 to 9).

For more information
Sandro Gori - EnginSoft
s.gori@enginsoft.com

by A. Bayatfar, R. Warnotte & P. Rigo
University of Liege, Belgium

Automated scantling optimization of ships' midship transverse frame in concept design phase

Case study on optimizing structural design of vessels

This paper concerns the scantling optimization of a vessel's midship transverse frame during the conceptual design phase. The main focus of the present research is to demonstrate an automated optimization process for a typical midship transverse frame of RoPax vessel which has been developed within the framework of EU HOLISHIP project (2016-2020). To this end, a number of existing tools along with their new script/batch-mode developments (namely STEEL®, a tool from Bureau Veritas for the structural strength assessment of primary transverse frames, and modeFRONTIER® as the optimization tool) as well as some new in-house tools/modules (e.g. Rule Infringement Indicator, Weight/CG Calculator) have been integrated under an automated iterative routine.

Shipbuilding and the shipping sector face increasing pressure to balance the conflicting needs and requirements of improving safety, reducing environmental impact, increasing flexibility for varying operational conditions, improving life cycle cost/performance, etc, within the context of a highly competitive market.

Meeting such significant challenges requires the use of a holistic, multi-disciplinary and multi-objective design optimization platform from the earliest design stages in the traditional ship design process. The creation of such a platform was to some extent addressed during some former research-based EU-funded projects such as IMPROVE (2006-2009) (for examples see the papers published by Rigo et al. (2010) and Klanac et al. (2011)),

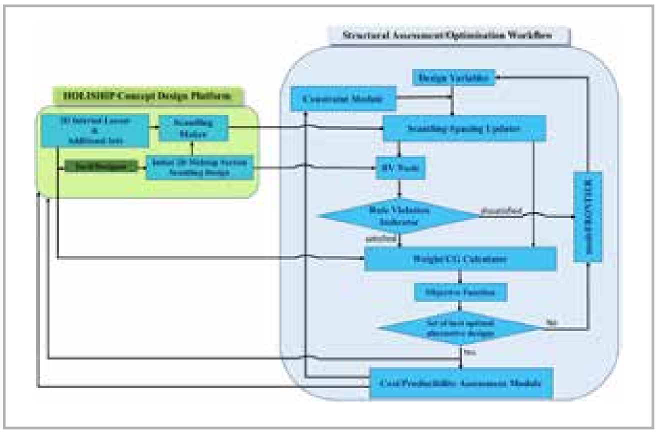


Fig. 1 - Integrated automated structural ship design assessment/optimization workflow for conceptual design purposes (Bayatfar et al. 2018; extended version).

and BESST (2007-2013) (for examples see the paper published by Bayatfar et al. (2013) regarding ship structure).

In continuation of preceding work undertaken, the EU launched a new R&D project in 2016, first initiated by Papanikolaou in 2010. A very large team of European partners set out to comprehensively develop the concept of a holistic approach to the optimization of ship design and has been collaborating to implement the approach within the context of the EU Horizon 2020 R&D project dubbed HOLISHIP, or the "Holistic Optimization of Ship Design and Operation for Life Cycle" (www.holiship.eu).

The HOLISHIP project addresses the different design steps from the concept ualization and contract design of vessels to virtual prototyping for design, and operational assessment. It aims to consider all relevant design aspects, namely energy efficiency, safety, environmental compatibility, production and life-cycle cost using holistic optimization platforms that aim to deliver the right vessel(s) for future transport tasks. To this end, a modern, flexible computer-aided engineering (CAE) environment, based on CAESES® (www.caeses.com) as the process integration and design optimization (PIDO) environment, is being used to integrate all the important disciplines of conceptual and contractual design under the greater umbrella of advanced parametric modeling tools to enable the parametric, multi-objective and multi-disciplinary optimization of ship products. (A detailed description of the project can be found in Papanikolaou (2019), while a quick overview can be gained from Harries et al. (2017) and Marzi et al. (2018).)

Fig. 2 - Midship transverse frame model built in STEEL® together with the loads in green

Recent feedback from shipyards indicates that efficient and effective weight reduction is one of the main technical requirements of the design process (Rigo et al. (2017)). To meet this need, within the EU HOLISHIP project, the following integrated workflow for automated structural design optimization (Fig. 1) has been developed to systematically serve application cases relevant to the conceptual design phase where simplified/innovative rules-based assessment methods/tools are necessary.

Continuing the work carried out by Bayatfar et al. (2019), the main objective of this research is to demonstrate an automated process for scantling optimization in a typical transverse frame in the midship of a RoPax vessel at the conceptual design phase. To this end, a number of existing tools along with their new scripts/ developments in batch mode (i.e. STEEL® (www.veristar.com) from Bureau Veritas to assess structural strength in primary transverse frames, and modeFRONTIER® (www.esteco.com) for optimization), and some new in-house tools/modules (e.g. Rule Infringement Indicator, Weight/Center of Gravity (CG) Calculator) have been integrated into an automated iterative routine.

In the automated optimization cycle, the structural weight is considered as an objective function to be minimized, and the constraints required under the applicable BV Class Rules for steel ships, and by the shipyard are also taken into consideration. The thickness of the web plate, the height of the web, the flange thickness and flange width of the primary transverse frames are the design variables considered. On the basis of this, the optimization results obtained are presented.

Case study

The present case study is for a RoPax vessel with its length, breadth, depth and speed equal to 217m, 32.2m, 9.65m and 24 Knots respectively. Fig. 2 below shows the typical transverse frame in the midship of this vessel together with the loads. The structure is made of mild steel (ST235) and high tensile strength steel (ST355) with a Poisson ratio and Young's modulus of 0.3 and 206 GPa respectively. The amount of load/wheel loads used on the loading decks in the hull were assumed to be equal to 43.2 KN/m, while the accommodation decks were loaded according to the applicable BV Class Rules.

Some details regarding the initial scantling design of the structural model can be seen in Table 1.

Optimization workflow/results

The scantling optimization workflow developed within the modeFRONTIER® environment is presented in Fig. 3, where the existing STEEL® tool along with its new scripting/batch-mode developments and numerous in-house tools/modules are integrated into an automated iterative routine for minimum effective structural weight while all constraints (BV Rules, technological and geometrical constraints …) are met. The integrated optimization cycle of the scantling is set up with 56 design variables (defined on the transverse frame of the midship hull, – and including the

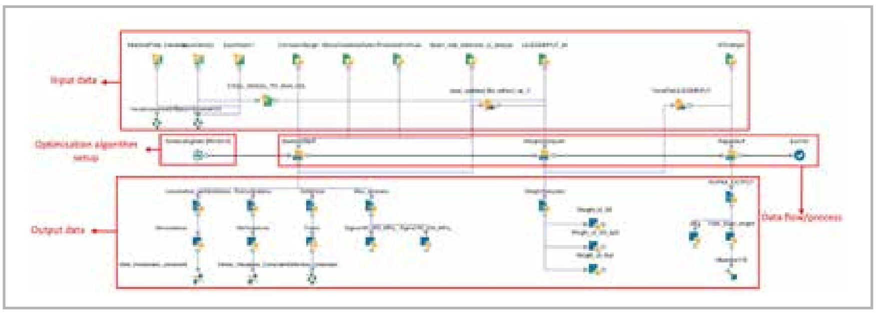


Fig. 3 - Automated scantling optimization workflow within the modeFRONTIER® environment.

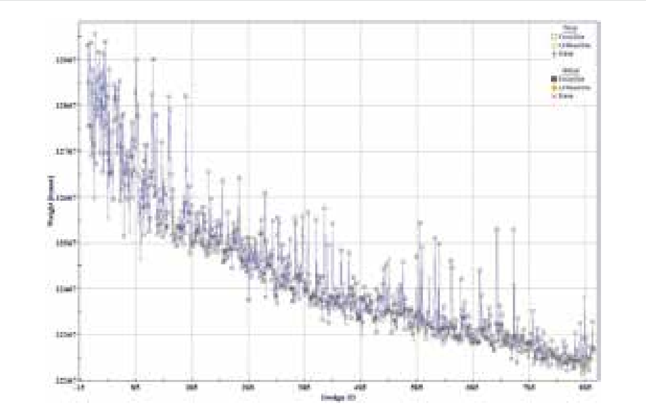


Fig. 4 - History of weight optimization.

thickness of the web plate, the web height, the thickness of the flange and flange width of the primary transverse frames).

From the design of experiments (DOE) techniques in modeFRONTIER®, the random algorithm was taken and an initial population of 29 designs was generated. From the library of optimization algorithms included in modeFRONTIER®, Multi Objective Genetic Algorithm II (MOGA-II) was chosen to determine which designs should be evaluated. The number of generations, the probability of crossover, the probability of mutations, and the number of evaluations were set to 30, 0.5, 0.1 and 5000 respectively.

Based on a feasible design achieved with the MARS2000®-based optimization workflow (see Bayatfar et al. 2019), the STEEL®-based workflow is called upon to perform the scantling optimization of the midship transverse frame. As an example, Fig. 4 shows the history of the structural weight of the RoPax vessel in a fully automated process, without any manual intervention through the graphical user interface.

The total calculation time per iteration in the STEEL®-based workflow, using a machine with an Intel® Core ™ i7 CPU 860 @2.80 GHz and RAM 12.0 Go., is, on average, 5.5 seconds. This means that the optimal feasible solutions can be obtained in less than two hours.

Table 1 provides a detailed comparison for two structural members i.e. bridge 1 and the lower side shell (see Fig. 2), between the initial scantling design with ID-0 and an optimal feasible alternative design with ID-882.

Conclusions

This project has been carried out within the framework of the research activity being undertaken by the European HOLISHIP Project. Considerable efforts have been made to successfully manage the correct connections and smooth the data exchange between the STEEL® BV tool and modeFRONTIER®, and the in-house developed tools in a fully automated manner. This project demonstrates a fully automated optimization process for the midship transverse frame of a RoPax vessel to achieve the minimum structural weight while meeting the BV rules/shipyard requirements. The development was carried out for general purposes and can thus be applied to other types of vessels.

One of the most important next steps to this development could be the consideration of multiple transverse frame sections along the length of the ship in order to more accurately estimate the structural weight of the vessel. This has been initiated by University of Liege (ULiege).

Acknowledgements

The authors wish to acknowledge the support provided by the H2020 project "HOLISHIP- Holistic Optimization of Ship Design and Operation for Life Cycle" under contract no. 689074.

The authors also appreciate all the support given to the current development by Shabeeb Fasil Ummathur and Mehmet Merdivenci (former students at ULiege).

For more information
Vito Primavera - EnginSoft
v.primavera@enginsoft.com

References

[1] Bayatfar, A. Amrane, A. Rigo, Ph. 2013. Towards a ship structural optimization methodology at early design stage. Int. J. of Engineering Research and Development. 9(6): 76-90.

[2] Bayatfar, A. & Rigo, Ph. 2018. HOLISHIP: Structural design optimization-concept phase. https://fairplay.ihs.com

[3] Bayatfar A., Warnotte R., Rigo Ph., 'Automated Structural Optimization of Ships' midship section in concept design phase', MARSTRUCT19, Croatia, 2019.

[4] BESST. 2007. Breakthrough in European ship and shipbuilding technologies. EU. http://www.besst.it/BESST/index.xhtml

[5] BV. 2018. Rules for the classification of steel ships (NR467). Part B: Hull and stability. http://erules.veristar.com/dy/app/bootstrap.html

[6] Harries, S. Cau, C. Marzi, J. Kraus, A. Papanikolaou, A. Zaraphonitis, G. 2017. Software platform for the holistic design and optimization of ships. STG Jahrbuch.

[7] HOLISHIP. 2016. Holistic optimization of ship design and operation for life cycle. EU. http://www.holiship.eu

[8] IMPROVE. 2006. Design of improved and competitive products using an integrated decision support system for ship production and operation. EU. https://cordis.europa.eu/project/rcn/81506/factsheet/en

[9] Klanac, A.& Varsta, Petri. 2011.Design of marine structures with improved safety for environment. Reliability Engineering and System Safety. 96: 75-90.

[10] Marzi, J. Papanikolaou, A. Brunswig, J. Corrignan, P. Zaraphonitis, G. Harries, S. 2018. HOLISTIC ship design optimization. 13th Int. Marine Design Conference.

[11] Papanikolaou, A. 2010. Holistic ship design optimization. Journal Computer-Aided Design, Elsevier. 42(11): 1028-1044.

[12] Papanikolaou, A. (Ed.) 2019. A holistic approach to ship design– Vol. 1: Optimization of ship design and operation for life cycle. Springer 978-3-030-02809-1.

[13] Rigo, P. Ehlers, S. Zanic, V. Andric, J. 2010. Design of innovative ship concepts using an integrated decision support system for ship production and operation. Brodogradnja. 61(4): 367-381.

[14] Rigo, Ph. Bayatfar, A. Buldgen, L. Pire, T. Echeverry Jaramillo, S. Caprace, J.D. 2017. Optimization of ship and offshore structures and effective waterway infrastructures to support the global economic growth of a country/region. Ship & Science Technology. 11:9-27.

ANSYS Solutions simplify additive manufacturing – the example of a drone

Concept becomes reality

Drones are one of the interesting products which require reduced weight and high strength. 3D printing, which is the most powerful manufacturing technique for making light structures, can be efficiently combined with topology optimization. For this project, Taesung S&E created a real drone with an organic, biological cellular lattice shape. ANSYS topology optimization and Discovery Live were used at the design stage to reduce weight and increase aerodynamic performance. ANSYS Additive Print was used to minimize the thermal deformation and residual stress, prevent blade crash and increase the manufacturing effectiveness. It also provides a compensated model for accurate dimensioning.

by ByungJu Yoo
DfAM Research Center

Objective

In general, additive manufacturing (AM) (also known as 3D printing) has special advantages for

- making structures with complex shapes that were impossible to make in the past
- reducing the number of parts as a one body

This project was initiated to prove the merits of metal additive manufacturing by creating a real product, a Racing Drone. Four detailed objectives were set:

1. Topology Optimization

- Improve the body rigidity as the first priority to maximize the responsiveness of the racing drone's flight controls for rapid maneuverability;
- Evaluation with the Light Weight Index to ensure maximum mass efficiency
- Organic and Biological design

2. Lattice structure

- Additional weight reduction by using a lattice structure
- Styling considerations

3. Titanium (Ti64)

- A difficult material to produce with conventional processing methods
- High corrosion resistance, light weight and high strength compared to steel
- No immunorejection by the human body (while this was not critical for the drone it is a consideration for the next project)

4. Feedback from AM field

- Easy to remove supports
- Support optimization

Development process

It was decided to create an easy process that any common designer, and not only a CAE specialist, could follow after taking some simple training. This was achieved by skipping the digital verification step usually conducted by experts.

ANSYS mechanical was used for the Design for Additive Manufacturing (DfAM) and topology optimization at the design stage.

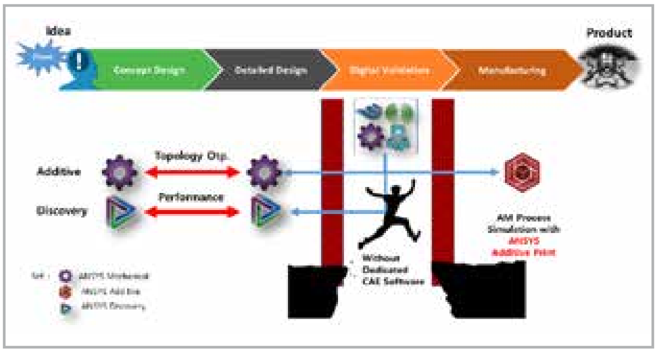


Fig. 1 - Development process

ANSYS Discovery Live was used for initial virtual validation and ANSYS Additive Print was used for the manufacturing process.

Topology optimization

There are many load cases and constraints which should be considered when performing topology optimization, such as forces, moments, vibration and so on. Fig. 2 shows the final topology shape which has an organic and biological appearance.

Based on the STL file from topology optimization, the mounting design and some modifications for the wire harness path were conducted directly in ANSYS Spaceclaim. To achieve a lighter design, a lattice structure was considered for the drone body center and cover. Important considerations were the reduction of mass and the ease of removing the supports.

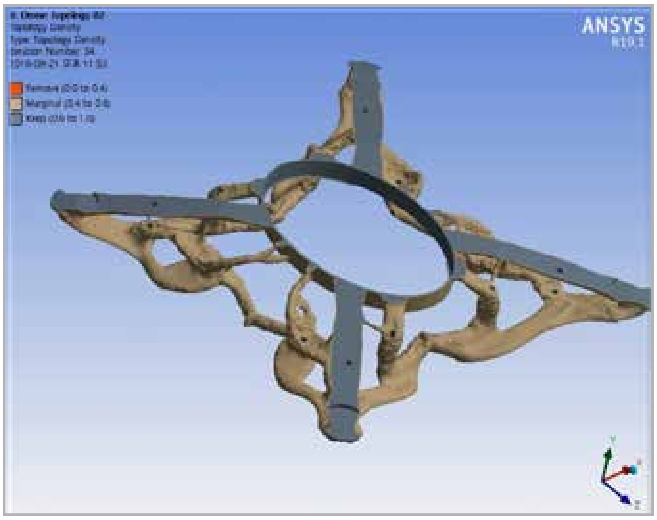


Fig. 2 - Topology optimization result

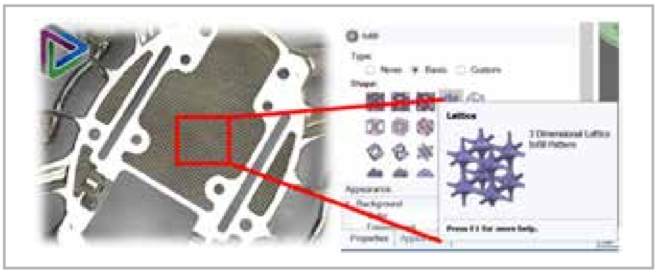
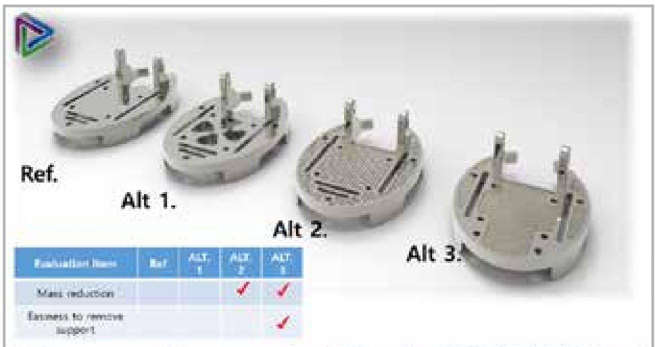


Fig. 3 - Lattice design function in ANSYS Spaceclaim



		Ref.	Trim	3D Lattice Infill Pattern	
			ALT. 1	ALT. 2	ALT. 3
Lattice	Length(mm)	-	-	1.6	0.8
	Thickness(mm)	-	-	0.4	0.2
	Fill Rate	-	-	14.7 %	14.7 %
Mass (g)		70.3	60.6	41.2	41.2
Mass reduction ratio		-	-13.8%	-41.4%	-41.4%

Fig. 4 - Alternative candidates and evaluation results of body design

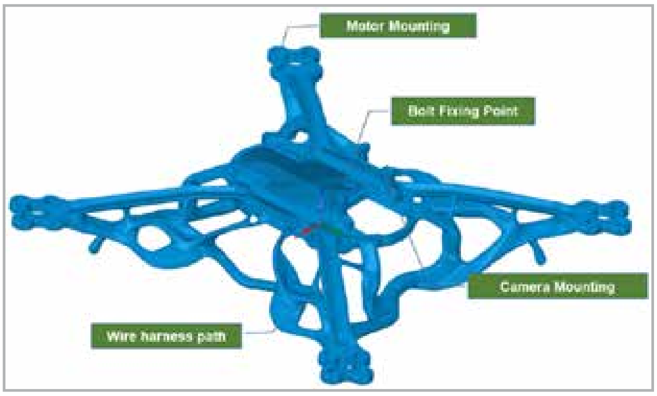


Fig. 5 - Final body design (AM result)

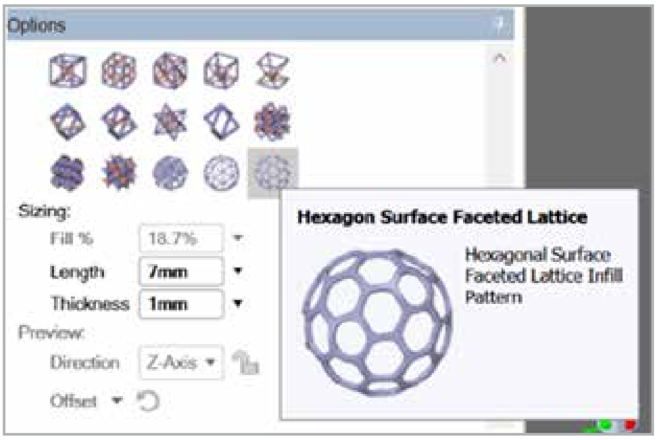
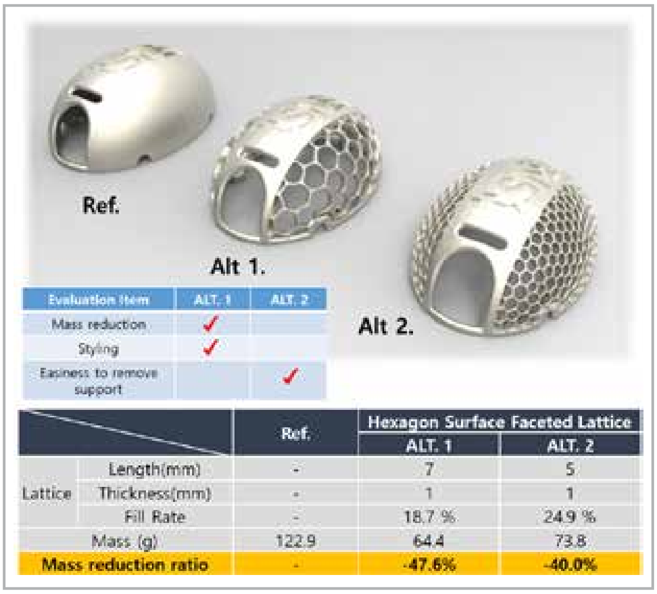


Fig. 6 - Hexagon surface function in ANSYS Spaceclaim



Evaluation Item	ALT. 1	ALT. 2
Mass reduction	✓	
Styling	✓	
Easiness to remove support		✓

		Ref.	Hexagon Surface Faceted Lattice	
			ALT. 1	ALT. 2
Lattice	Length(mm)	-	7	5
	Thickness(mm)	-	1	1
	Fill Rate	-	18.7 %	24.9 %
Mass (g)		122.9	64.4	73.8
Mass reduction ratio		-	-47.6%	-40.0%

Fig. 7 - Alternative candidates and evaluation results of the cover design

Compared to the reference model of the body, alternative 2 (Alt. 2) and alternative 3 (Alt. 3) show similar weight reduction -- about -41%. But body design alternative 3 (Alt. 3) was selected for its convenience of removing the supports. Next, a hexagon surface faceted lattice structure was applied to reduce the cover weight. Even though candidate Alt 1 offered good weight reduction, Alt 2 was selected because it offered the same convenience of support removal as the body design.

If the hole size were small, the structure could be manufactured without supports. But if the hole size is larger than a specific dimension, many supports are required to prevent deformation during AM processing.

In Fig. 8, one can see many supports directly attached to Alt 1's hexagonal surface. But, by using Alt 2, we could manufacture the entire lattice structure at once without any inside supports attaching to the hexagonal surface.

Fig. 9. was made with ANSYS Spaceclaim's Keyshot module and enabled the design and styling of the drone to be finalized. The next steps were to check the strength and the aerodynamic performance and to create the product without any defects (deformations, cracks, etc.)

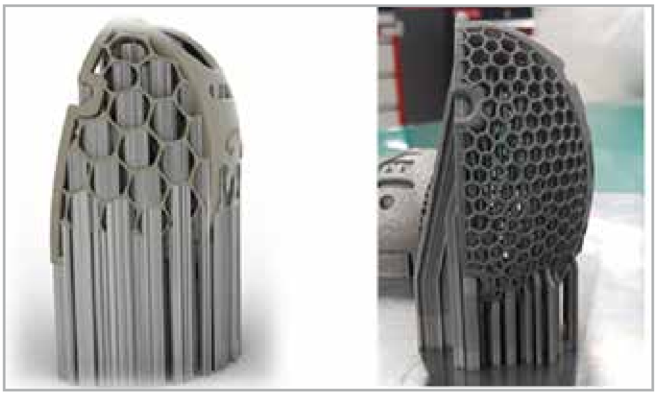


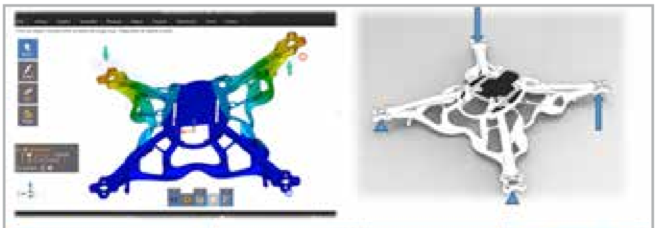
Fig. 8 - Supports in Alt 1 (left) and Alt 2 (right)

Fig. 9 - Rendering with ANSYS Spaceclaim

Virtual validation

The simplest and most common way to check structural safety is with a static stiffness analysis. Among several static stiffness analyses, we conducted a torsional stiffness analysis (also known as rigidity) which is usually used for bodies that rapidly accelerate/decelerate, like racing drones. In general, when the drone changes direction rapidly, a low rigidity drone would distort.

The purpose of the topology optimization was to reduce the weight while increasing the stiffness, but sometimes these two requirements conflict. Using a light weight index (mass/torsional stiffness) is one way to evaluate how well the optimization was done. Sometimes area (A) is used to compare the model size: the lower the index value, the better the design. The model with topology optimization only showed no weight/



Item	Ref.	Topology	Topology & Lattice
Mass (M)	1 (295g)	1.2 (354g)	1.09 (323g)
Size (Diagonal Length)	1 (210mm)	1.2(260mm)	1.2(260mm)
Mass/Size	1.00	0.97	0.88
Torsional Stiffness(C_t)	1	2.2	1.7
Light Weight Index	1.00	0.44	0.52

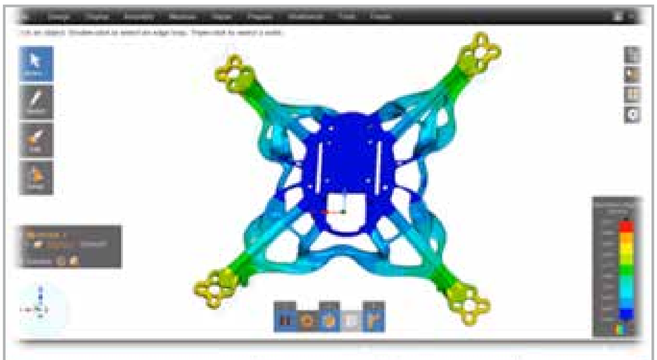
$$\therefore Light\ Weight\ Index = \frac{M}{C_t \times A}$$

Fig. 10 - Torsional stiffness and light weight index

size reduction, but the stiffness doubled, resulting in an index of 0.44. After applying the lattice, weight was reduced by -12% and a torsional stiffness increment of 70% was achieved (index=0.52). This meant the design was almost two times better than the reference model.

It is important to emphasize how easy this simulation process is. With ANSYS Discovery Live, it only took 1 minute from importing the STL file to obtaining the post-stress and deformation results.

Next, the dynamics stiffness (modal analysis) was checked.



Item	Ref.	Topology	Topology & Lattice
1st Mode (Hz)	167	250	214
Ratio	1	1.5 ✓	1.3

Fig. 11 - Dynamic stiffness

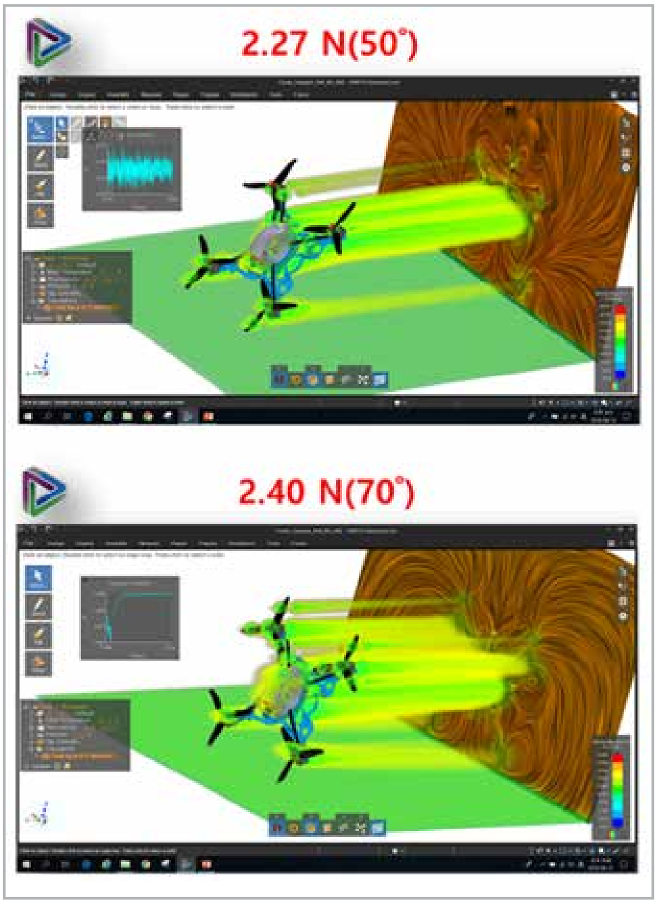


Fig. 12 - Aero-dynamic performance (drag force)

In conclusion, the topology model was superior to the additional lattice mode for both static and dynamic stiffness. We selected the "Topology and Lattice" application model just for an experimental application of the LATTICE structure.

Finally, an external flow analysis was conducted in ANSYS Discovery Live to check the drag force at several attack angles. This analysis was completed in 10 (s) without any additional geometry modification or meshing operations.

AM build

ANSYS Additive Print was used to pre-check possible thermal deformation, blade crash, high strain severity and residual stress during the AM process. Blade crash means a Z-directional deformation. If the value is large, the product will be hit by the recoater blade and the product could be broken during processing. If strain severity is high, a crack could easily occur. If residual stress is high, there could be durability problems during use. The After cut off displacement plot shows the product's deformation after cutting the supports this is related to the product's precision. All of these elements can be checked in ANSYS Additive Print immediately.

In addition, ANSYS Additive Print offers optimal support design which minimizes previous problems.

The role of the supports is to support the product and prevent thermal deformation during AM processing. They can also serve as thermal paths to the base plate to facilitate dissipation of high temperatures. On the

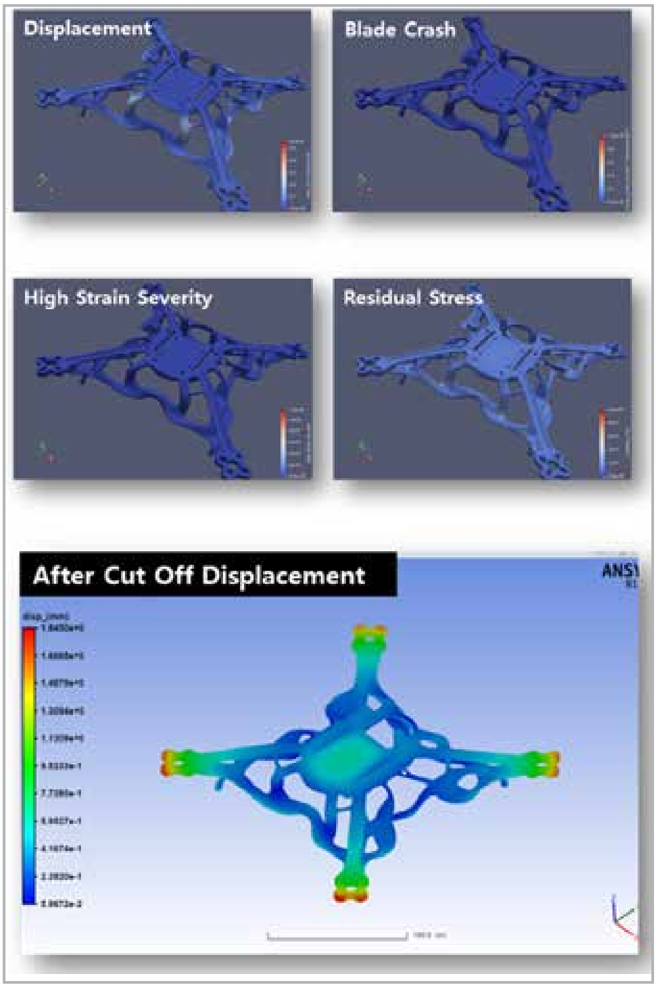


Fig. 13 - Problem estimation during AM processing

other hand, many supports are difficult to remove and increase product costs, material and time wastage. So, the most effective support locations and methods must be selected.

Alt 1 is a general 45-degree type for reducing the supports, and Alt 2 is a parallel type with many check points. We selected the Alt 2 type after considering the manufacturing speed, the surface quality and the price of titanium powder. Fig. 14 below shows the supports created with the thick wall method as suggested by ANSYS Additive Print. The black-looking area is the area of most inter-supports due to the lack of variation in thermal stress.

In summary, ANSYS additive print generates an optimal support design based on a prediction of thermal deformation, unlike other programs that consider geometry only.

Finally, the drone body was successfully created. Fig. 15, top, shows the products taken directly from the Metal AM and the image below shows it after removing the supports and thermal treatment.

The cover was created with two different orientation angles. One was for minimizing the number of supports and the other was for the surface finishing. Both were successfully produced.

Fig. 17, left, is a rendering of the drone model in ANSYS SpaceClaim's Keyshot. The concept was developed with computer simulations considering topology optimization, lattice design and AM processing

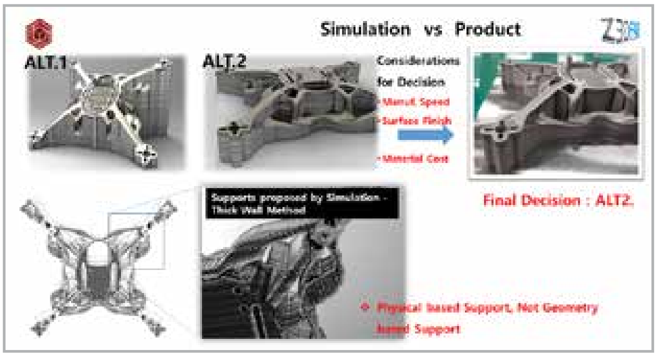


Fig. 14 - Parallel dupport (top), thick wall method (below)

Fig. 15 - Before and after removing the supports (body)

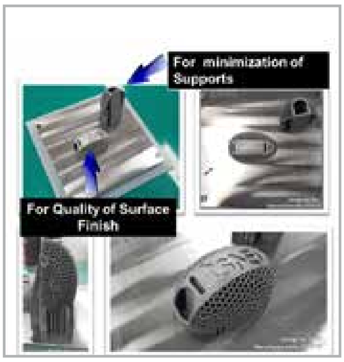


Fig. 16 - Cover

in ANSYS Product. On the right is the final product. The entire process was completed in two weeks. In the meantime, we created a plastic version for possible assembly.

Isn't it amazing?

This Drone project made us realize that all design processes

Fig. 17 - Before and after removing the supports (body)

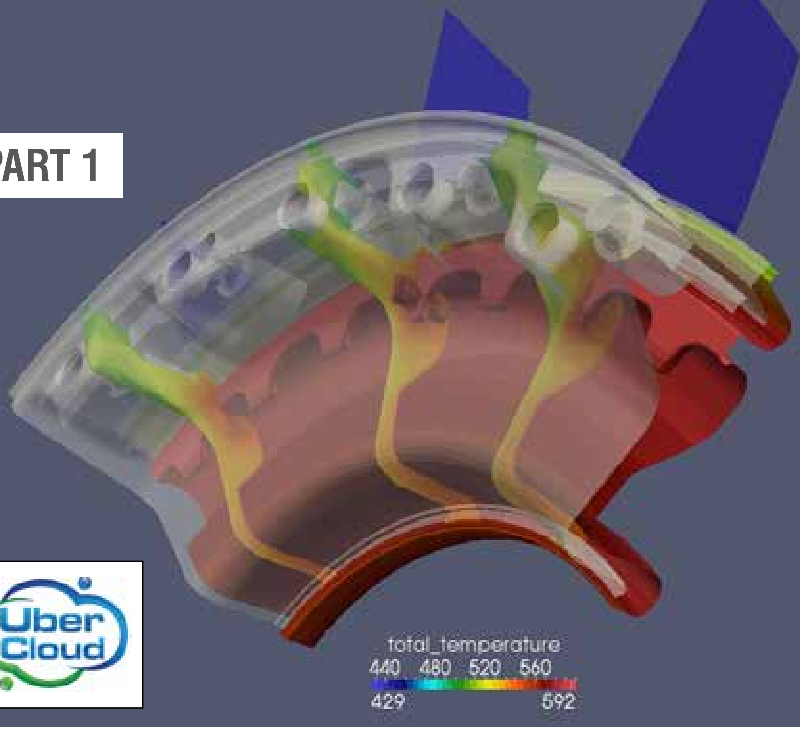
can be undertaken in a CAE-environment and that concept design can create real designs without any complex or real tests. ANSYS helped us to realize all this process.

The real flying drone we made can be seen at the link below.
www.youtube.com/watch?v=WAjpd_0jpwE

For more information
DfAM Research Center - Tae Sung S&E
dfam@tsne.co.kr

PART 1

Engineering simulations using ANSYS in the cloud



By Wolfgang Gentzsch
UberCloud

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

Enterprises already started using cloud resources in 2006 when Amazon turned one of its computer farms into a service and offered it to the rest of the world. Thirteen years later, in 2018, Amazon Web Services (AWS) generated $25.7 billion in revenues, a 47% increase over 2017. However, engineering companies only began slowly becoming aware of this promising novel service computing model around 2012.

Granted, the major reasons for this hesitant start were the large number of additional roadblocks and concerns to be found in the engineering world, such as the then lack of security, traditional software licensing models, large data transfers, the engineer's fear of losing control over their assets, and even worse, internal resistance from management and IT departments who often barricaded themselves behind 30-year-old compliance regulations.

In 2012, we started a series of engineering simulation cloud projects, later known as UberCloud Experiments, which moved an engineer's simulation workflow to the cloud with the goal of analyzing the benefits and challenges, and publishing the lessons learned with recommendations for the engineering community [1]. Fast forward seven years, and we have now performed over 210 high performance computing (HPC) cloud experiments and published more than 100 case studies, 25 of them based on ANSYS software such as CFX, Fluent, Mechanical, LS-DYNA, HFSS, and Discovery Live. The interested reader can download the ANSYS and other case studies [3]. We have also been able to remove almost all the roadblocks by developing a software technology which hosts the engineer's complex workflows in an isolated software container that sits on a dedicated HPC resource in any cloud – public or private, hosted, and hybrid.

We have been able to objectively measure the progress of cloud computing over the past seven years. Looking back at our first 50 cloud experiments in 2012 - 2013, 26 projects failed or weren't completed because of the roadblocks mentioned above, while the average duration of the successful ones was about three months. Five years later, in 2018, none of our last 50 cloud experiments failed, while the average duration of these experiments was just about three days, which included defining the engineering application case, preparing and accessing the application software like ANSYS in the cloud, running the simulation jobs, evaluating the data via remote visualization, transferring the final results back onto premises, and writing the case study.

"The company was interested in reducing the solution time and increasing mesh size to improve the accuracy of their simulation results without investing in a computing cluster that would be utilized only occasionally."

This article is the first part of a series that will discuss a selection of case studies dealing with engineering projects based on ANSYS CFX and Fluent. The articles will briefly introduce the engineering use case, describe the cloud implementation, the benefits and challenges, and will conclude with the lessons learned and the recommendations.

CFX simulation of a flash dryer with hot gas to evaporate water from a solid

We selected this case study for historic reasons: it was our very first engineering cloud project based on ANSYS software (in this case CFX 14) and dates back to 2013! In the second piece of this case study we will briefly summarize our most recent project with FLSmidth, which we have just completed.

The 2013 project team consisted of Sam Zakrzewski from FLSmidth, founded in 1882 with its headquarters in Copenhagen, Denmark and with 11,300 employees in 60 countries, the company is a leading supplier of complete plants, equipment and services to the global minerals and cement industries.; and the French company Bull (acquired by Atos) serving as the cloud resource provider supported by Science & Computing (also acquired by Atos) from Tübingen/Germany.

FLSMIDTH use case

The company was using computational fluid dynamics (CFD) multiphase flow models to simulate a flash dryer designed for a phosphate processing plant in Morocco. The dryer would take a wet filter cake and produce a dry product. Increasing plant sizes in the cement and mineral industries demanded that the existing designs be expanded to fulfill customers' requests.

At the time of the project, the multiphase flow simulation took about five days on FLSmidth's local infrastructure (workstations, Intel Xeon X5667, 12M Cache, 3.06 GHz, 24 GB RAM) to generate a realistic particle loading scenario. The differential equation solver for the Lagrangian particle tracking model required several GBs of memory while the simulations used 1.4 million cells, five species and a time step of one millisecond for a total time of two seconds. The cloud solution, instead, allowed the models to be run much faster, increasing the turn-around time of the sensitivity analyses and reducing the time to customer implementation. It also enabled the focus to become the engineering aspects instead of valuable time being wasted on IT and infrastructure problems.

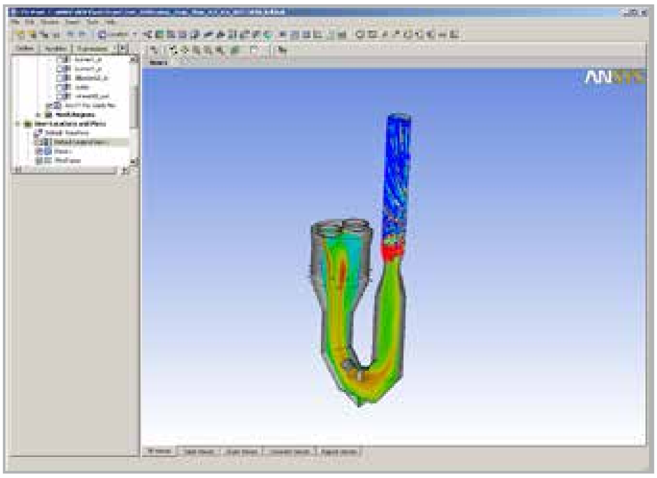


Fig. 1 - Flash dryer model viewed with ANSYS CFD-Post.

FLSmidth was interested in reducing the solution time and increasing the mesh size to improve the accuracy of the simulation results generated, without having to invest in purchasing its own computing cluster that would be utilized only occasionally. The model selected for the project posed a challenge due to the scalability of the problem which required an increasing number of cores. To address this, Bull's Xtreme Factory (XF) team integrated ANSYS CFX into its web user GUI to simplify the data transfer and then ran the simulation on dedicated XF resources with up to 128 Intel E5-2680 cores. The numerically intensive tasks were performed in parallel, and only the administrative tasks such as simulation control, user interaction, and input/output were performed in sequential mode by the master process. While the final runtime of this ANSYS CFX simulation job still took about 46.5 hours, FLSmidth's primary goal of running the job in one to two days, was successfully achieved.

FLSmidth, five years later

In September 2018, Sam Zakrzewski from FLSmidth approached UberCloud again to perform an extensive Proof of Concept to evaluate whether the timing was right to consider moving its HPC simulation workload onto the Microsoft Azure Cloud.

Zakrzewski's team, which is now distributed between locations in Copenhagen, India, and Brazil, drafted a list of their actual requirements with a two-year roadmap of moving different simulation scenarios using ANSYS CFX and ESSS Rocky onto the Microsoft Azure Cloud. FLSmidth currently has its own on-premises Haswell-based HPE cluster with 512 cores and Infiniband FDR for its 20 habitual users. In the next step, it wishes to increase its user base and upgrade the on-premises environment with cloud bursting for mission-critical applications in CFD (multi-phase, combustion) using ANSYS CFX and Fluent, STAR CCM+, and ANSYS Mechanical (static, thermal, modal, fatigue). In addition, the company is applying the discrete element method (DEM) to simulate granular and discontinuous materials with ESSS Rocky. The first phase of this project has just been successfully completed; the second phase will focus on the creation of a global cloud solution to include all the company's CAE engineering teams worldwide.

> *"The extra number of cores we got access to in the cloud helped us extremely by reducing the time needed to run all our simulations."*

Foro Energy used ANSYS Fluent to analyze geo-thermal perforation

Foro Energy commercializes high power lasers with the capability and hardware platform to transmit over long-distance fiber-optic cables to enable step-change performance in applications to drill, complete, and workover wells in the oil, natural gas, geothermal, and mining industries. Sabalcore Computing provides HPC cloud services for government, commercial industry and academic institutions with target implementations in the life sciences, weather modelling, engineering and design, financials, and oil and gas.

Foro Energy use case

This project studied geo-thermal perforation using ANSYS Fluent. The scope was to characterize the flow propagation through the test setup and recommend the design changes needed to improve it. The test setup was submerged inside water at a given pressure, i.e. the surrounding container was filled with water at the given ambient pressure. The goal was to hit a target with the laser whose dispersion was reduced by the medium. A fluid-like nitrogen or liquid CO_2 was used for the guided flow. The path travelled by the laser was cleared of the water medium by using the guided fluid jet at a high flow rate. The goal of this study was to determine the optimal flow rate of the guided fluid required to ensure that the laser path was cleared of water and was filled with the guided fluid, which serves as a better medium for laser propagation compared to water.

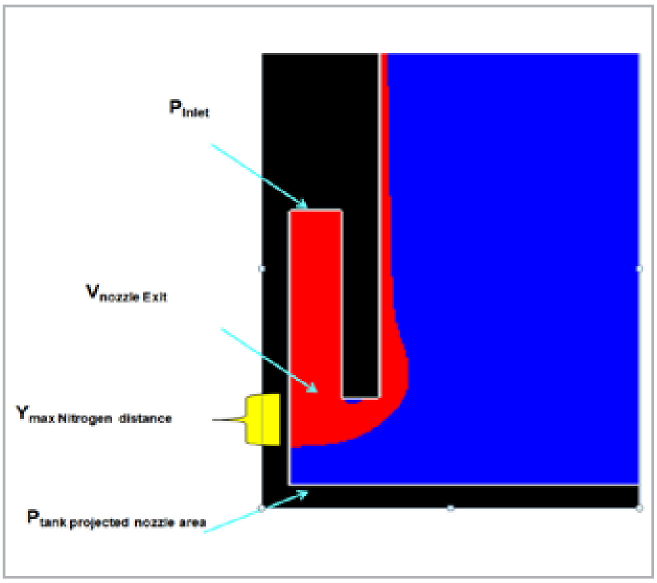


Fig. 2 - Sample steady state of guided fluid (nitrogen) phase distribution for all inlet flow rate conditions.

Modeling

To determine an optimal flow rate, a numerical model is created and solved for different parameters. The first step is to create a solid model for the setup, which represents the actual laser operating system together with the walls and the bottom of the tank/well used in the experiment, or in actual field operation. This solid model is commonly generated in SolidWorks. From the detailed solid model of the actual laser operating system, a representation of the model is created and imported into ANSYS Design Modeler for further processing, to clean and smooth the surfaces, and to patch any discontinuities which may have arisen during importing.

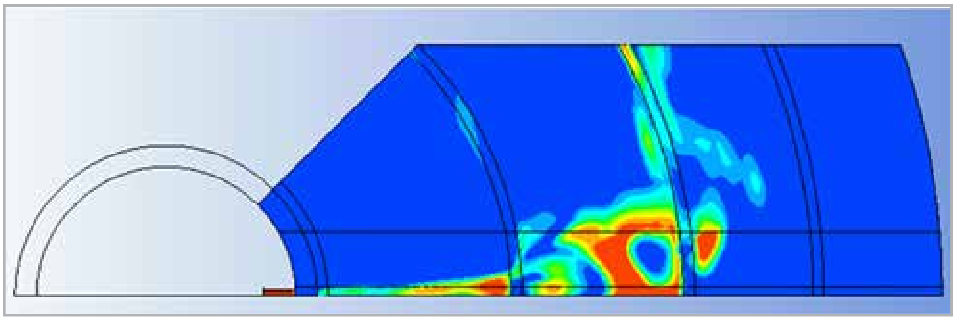


Fig. 3 - Flow field inside the domain (cross section of the 360o slot) as the nitrogen gets distributed.

The fluid domain is then transferred to the mesh generator where an optimal mesh is generated. The final mesh is transferred to ANSYS Fluent for CFD analysis. Boundary conditions are applied at all faces of the domain, and pressure field distribution is applied to the entire outlet boundary surface by taking into consideration the effect of gravity. All the outlet boundary surfaces also have water as the back-flow condition. Based on the Reynolds and Prandtl number of the flow, appropriate turbulence fluid model discretization schemes and the time iteration method were selected.

A steady state model is then solved with ANSYS Fluent to understand the development of the nitrogen phase. Based on the total number of elements, and the expected computational cost, the simulations are usually performed on a local desktop machine with 8 cores. Fluent uses Intel MPI to divide the model equally across all the cores.

For large numbers of elements, the Sabalcore Linux servers are used to run the simulations on up to 32 cores using Infiniband interconnect and ssh to communicate between the nodes. After the successful completion of the simulation, the data is transferred to CFD-Post for post processing and to analyze the results, which could either be performed directly in the Sabalcore HPC Cloud or locally.

In the sample case in Figure 2, various boundary conditions and the monitoring zones are displayed. P_{inlet} is the inlet boundary

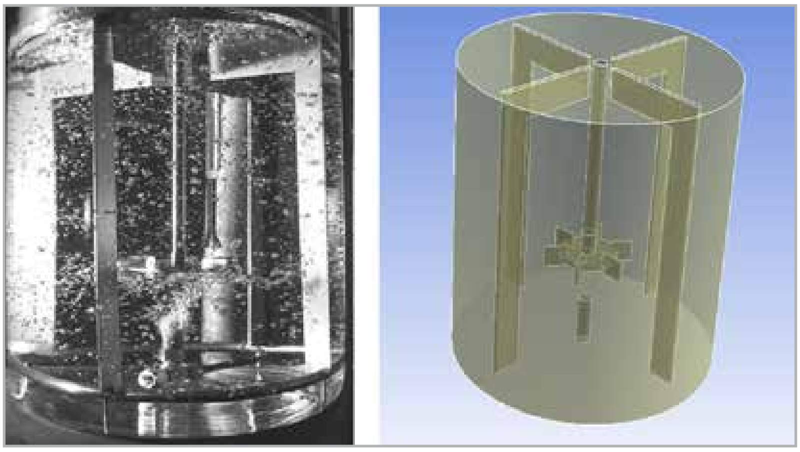


Fig. 4 - 194L Tank used for the experiments (Marko Laakkonen) and the representative CFD Model.

condition where a fixed flow rate of nitrogen is applied. The V_{nozzle} exit is the surface zone at the nozzle exit where the velocity of flow is computed, and this value is then compared with the theoretical value. The P_{tank} projected nozzle area is the projection of the nozzle on the wall surface/target where the pressure value is computed during the simulation. The Y_{max} nitrogen distance is the maximum distance the nitrogen is able to penetrate inside the water domain for the given inlet flow rate. The red color is the guided fluid (nitrogen in this specific case) coming out of the nozzle and the blue color is the water body.

Challenges

Most of the challenges lay in the CAD/CAE process of setting up the geometry and physics, e.g. to find the perfect mesh for the geometry and the best-suited physical parameters, such as for the boundary and the turbulence. The team did not face any challenges with accessing and using Sabalcore's cloud computing resources.

Conclusions

The team studied the effect of the inlet guided fluid flow fields for different scenarios, the optimal inlet flow rates, and the cases where the guided fluid, primarily nitrogen, was used to clear the path in the water medium. They also studied the cases where nitrogen flow was used to prevent the debris and particles from entering the nozzle, and the effect of particle diameter and particle density on the penetration length of the particles.

ANSYS Healthcare – establishing the design space of a sparged bioreactor

Scaling-up pharmaceutical laboratory mixers to a production tank is not a trivial task. It requires a thorough understanding of complex turbulent and multiphase processes that impact oxygen mass transfer. The interplay between the geometric design of the tank and the tank operating parameters are critical to achieving good mixing, especially at (larger) production scales. In an effort to improve the process understanding, international regulators suggested a Quality by Design (QbD) approach to process development and process control. In the QbD framework, significant emphasis is placed on the robust characterization of the manufacturing processes by identifying the engineering design space that ensures product quality. There are various geometric and operating parameters that influence oxygen mass transfer scale-up from lab scale to production scale. Understanding the effect of these parameters can result in a more robust design and the optimization of the bioreactor processes.

ANSYS use case

The main objective of this study was to understand the impact of the agitation speed and the gas flow rate on the gas holdup and mass transfer coefficient, which are two critical parameters that help process engineers understand mass transfer performance. The simulation framework was developed and executed on Microsoft Azure Cloud resources running ANSYS Fluent in the UberCloud software container. This solution provided a scalable platform to achieve sufficient accuracy while optimizing the solution time and resource utilization.

Process overview

The stirred tank is agitated by a 6-bladed Rushton turbine blade to disperse the air bubbles generated by the sparger. Four custom baffles are included to prevent vortex formation. The experimental conditions and results were taken from the extensive study performed by Laakkonen [2]. A full 3D model of the tank was considered for this study, and was meshed using polyhedral elements.

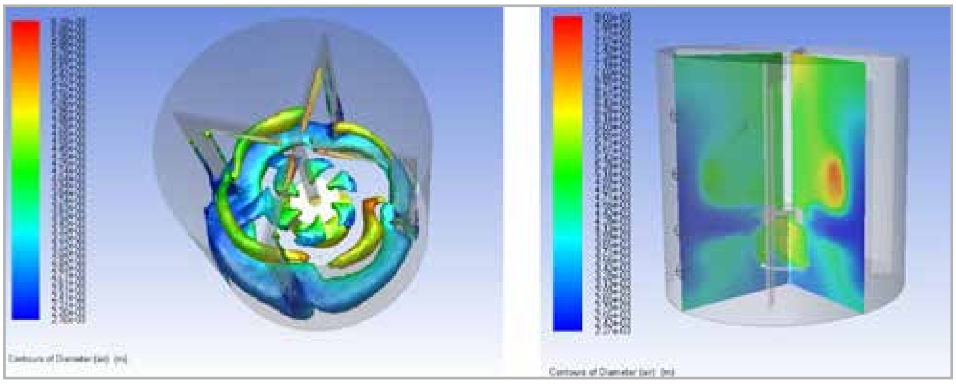


Fig. 5 - a) Iso-surface of gas volume fraction colored with bubble diameter; b) Contour plot of bubble size distribution.

The Eulerian multiphase model was used for simulating the two phases: water and air. The population balance model with the quadrature method of moments (QMOM) was used to simulate the bubble coalescence and breakup processes. The Ishii-Zuber drag model was used to account for the momentum exchange between the water and the air bubbles. For bubble coalescence, a model based on the Coulaloglou-Tavlarides model was used, while the breakup model was based on the work of Laakkonen.

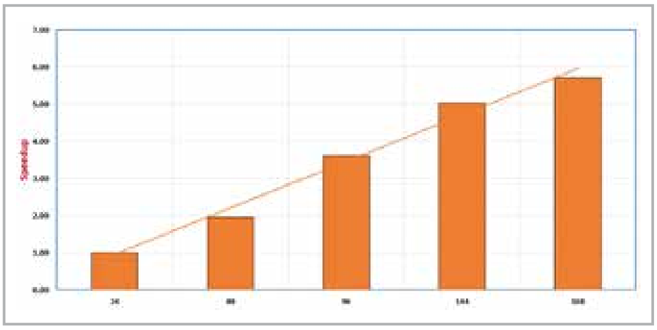


Fig. 6 - Speedup of 688K polyhedral mesh at different CPU cores.

It was observed that non-drag forces did not significantly impact gas holdup and mass transfer. A zero-shear boundary condition was applied for the water phase at the upper free surface, and a degassing boundary condition was used to remove the air bubbles. A design of experiments (DOE) study was performed with the agitation speed and the gas flow rate as the input parameters and with the volume-averaged mass transfer coefficient as the output parameter. ANSYS Workbench with DesignXplorer was used to run the DOE and to study the bioreactor design space.

Results

As shown in Figure 5, air bubbles undergo breakup near the impeller blades and coalesce in the circulation regions with low turbulent dissipation rates. This leads to varying bubble size throughout the tank. Since the interfacial area depends on the bubble size, bubble size distribution plays a critical role in oxygen mass transfer.

To study the design space of the bioreactor, a DOE study was performed to generate the response surface for the average mass transfer coefficient, which showed that agitation speed has a greater impact on the mass transfer coefficient versus gas flow rate. Therefore, studying the design space with several input parameters provides an opportunity to optimize the operating conditions to identify a safe operational range for the bioreactor.

HPC performance benchmarking

We used the cloud resources in Microsoft's Singapore data center because it is relatively close to the ANSYS office in Pune, India.

The experiment start date was 27th December 2017, and the experiment finish date was 30th January 2018. The simulations started on 1 node (16 cores) and the last run was executed on 16 nodes (256 cores).
The instance node type was a Standard _ H16r with FDR InfiniBand (56Gbps bandwidth), and Azure compute instances of 16 CPU cores (Intel(R) Xeon(R) CPU E5-2667 v3 @ 3.20GHz) with 112GB of memory.

The software used to simulate the gas sparging process was ANSYS Workbench with FLUENT hosted in an UberCloud HPC container that is fully integrated with the Microsoft Azure cloud platform.

Figure 6 summarizes the scalability study, which was based on the 688K polyhedral mesh. As can be seen, the solution speed scales close to linear up to 168 CPU cores. When using 168 cores, each simulation takes less than an hour, making it possible to run the entire design space of the bioreactor in less than 24 hours.

Benefits

1. The HPC cloud environment with ANSYS Workbench/ FLUENT and DesignXplorer streamlined the process of running the DOE with drastically reduced process time.
2. Simulating ten design points and generating the response surface took only 24 hours of run time with 144 CPU cores. This means that design engineers can quickly execute DOE analyses to study the scale-up behavior of their bioreactors.
3. With the use of VNC controls in the web browser, HPC Cloud access was very easy requiring minimal installation of any pre-requisite software. The entire user experience was similar to accessing a website through a browser.
4. The UberCloud containers helped smooth the execution and provided easy access to the server resources. The UberCloud environment integrated with the Microsoft Azure platform proved to be powerful because it facilitated the running of parallel UberCloud containers while providing a dashboard in the Azure environment, which helped to view the system performance and usage.

Conclusion and recommendations

The Azure/UberCloud HPC provides a very good solution for performing advanced computational experiments that involve high technical challenges with complex geometries and multi-phase fluid flow interactions that would not typically be solved on a normal workstation, reducing the time required to establish a two-parameter design space for a bioreactor to a single day and thus permitted the completion of the project within the stipulated time frame.

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

References

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

[2] Marko Laakkonen, Development and validation of mass transfer models for the design of agitated gas-liquid reactors, https://pdfs.semanticscholar.org/c6bd/d98a364a73fecb84468da9352659e475344d.pdf

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

Using simulation to design energy harvesters for power sensors

Development of the models for FSI simulation

The following article is the second part of a series. The first part ("Harvesting energy to power sensors that monitor air pollution" by Taurisano, Rizzi and Tafuro, published in ES Newsletter, year 15, n°4 winter 2018) described the project scope and the challenges being faced by the project partners in developing an energy harvesting device for Internet of things (IoT) nodes.

Project REM

Energy harvesting is the process of capturing small amounts of environmental energy that can be used to perform helpful tasks. Interest in energy harvesters has rapidly increased over recent years due to their wide range of applications: sensors for wearable systems, low-consumption actuators, wireless transmitters and other IoT devices all need to be energetically autonomous for their capabilities to be exploited.

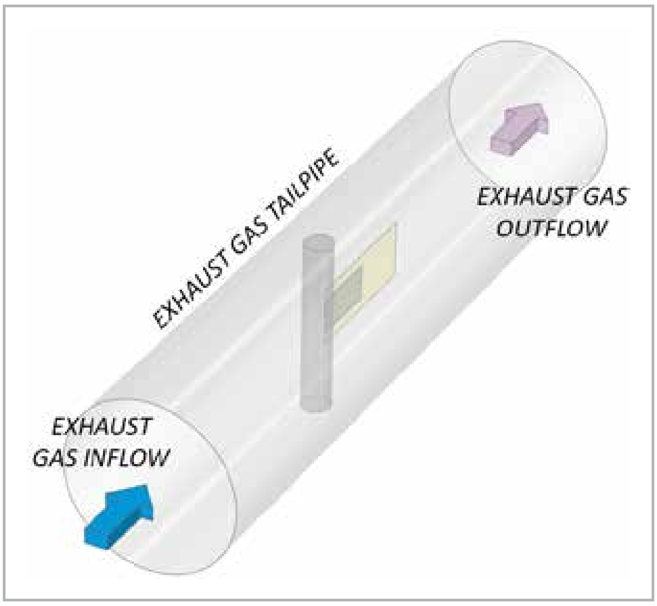


Fig. 2 - Example of the system layout

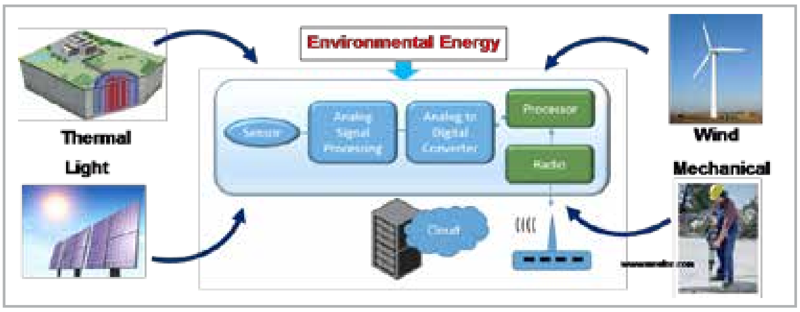


Fig. 1 - Harvesting environmental energy

To this end, EnginSoft, the Istituto Italiano di Tecnologia (IIT) and WebElettronica have been partnering in a project to develop an innovative energy harvesting system to recover mechanical energy from the exhaust gases of internal combustion engines using micro electro mechanical systems (MEMS) materials with piezoelectric properties to generate electric power for COx concentration sensors. This signal should

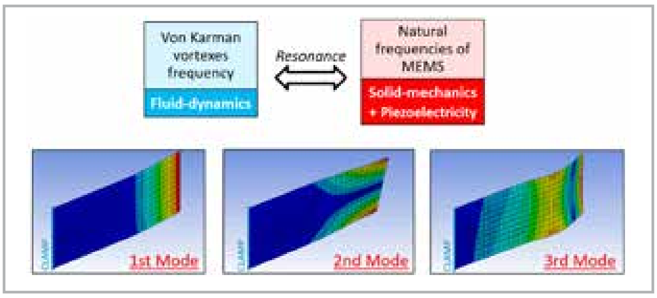


Fig. 3 - First mode shapes of the flag and the resonance effect

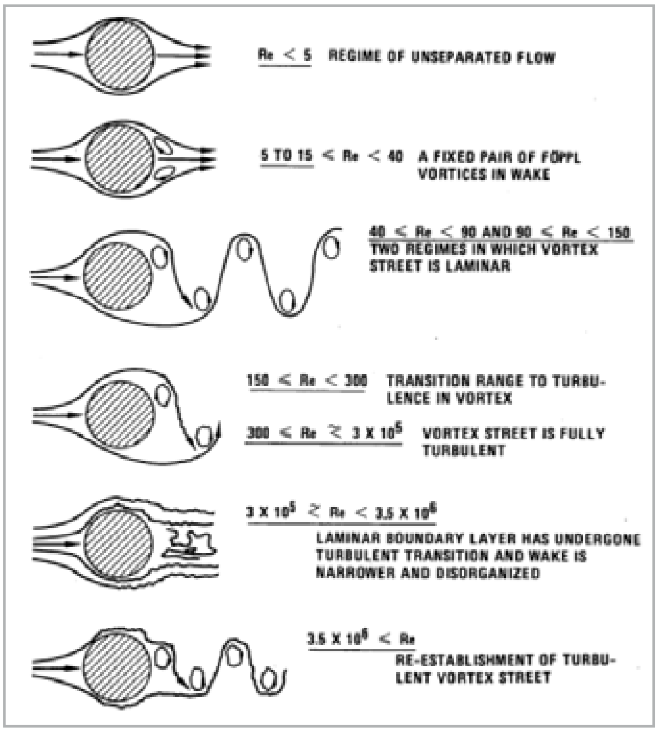


Fig. 4 - The Reynolds number dependency of Von Kármán vortices

be delivered to a motherboard located few meters away via the Bluetooth® protocol.

EnginSoft has been involved in the project to also develop the simulation process for the MEMS piezoelectric actuator. Starting from scratch, and by building a digital twin of the device as it was assembled in a wind tunnel, all possible configurations of the system layout have been studied to find optimal solutions for the entire operating range.

Two levels of system simulation are required:

- Fluid-structure interaction (FSI) analysis to accurately simulate the device's operation and determine how to exploit the Von Kármán vortex shedding phenomenon to generate electric current from the gas exhaust during operations.
- A digital twin using the reduced order models (ROMs) algorithm (trained with the results from the FSI simulations) for design of experiments (DoE) and layout exploration allows thousands of virtual tests to be run in a short computational time to find the best configurations for every operating condition.

Co-funding Program: "Horizon 2020 – PON 2014/2020"

Partners:

- EnginSoft (project coordinator) www.enginsoft.com
- Center for Biomolecular Nanotechnologies of the Istituto Italiano di Tecnologia (IIT) www.iit.it/centers/cbn-unile
- Web Elettronica www.webelettronica.com

This article will illustrate the development of the methodology for the FSI models, while the other project tasks will be discussed in future editions of the EnginSoft Newsletter.

Exploiting the mechanical load of vortex-induced vibrations

A bluff body hit by a gas flow generates alternating vortices, also called Von Kármán vortices. The low-pressure core that develops within these structures creates significant suction. If a flexible flag is placed downstream of the body, the suction is able to deflect the flag along its length.

IIT has developed a technology to create very thin and flexible flags by depositing materials with piezoelectricity properties.

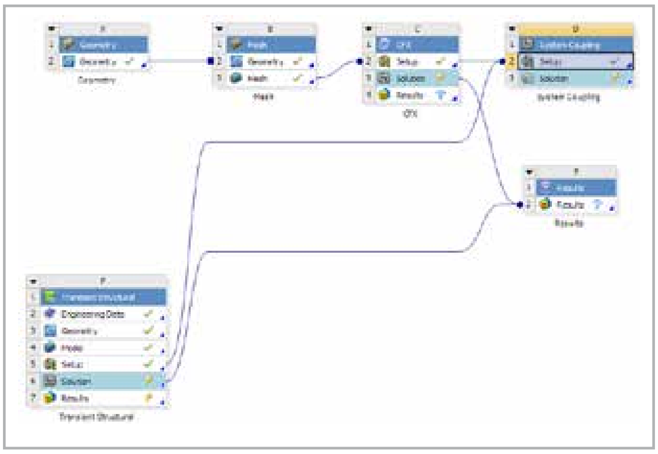


Fig. 5 - The fluid-structure interaction (FSI) simulation workflow in ANSYS Workbench

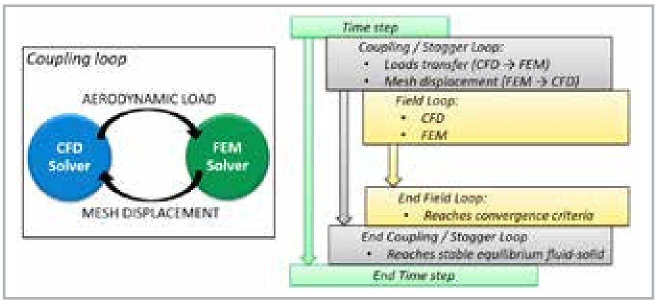


Fig. 6 - Numerical coupling using the implicit algorithm in FSI

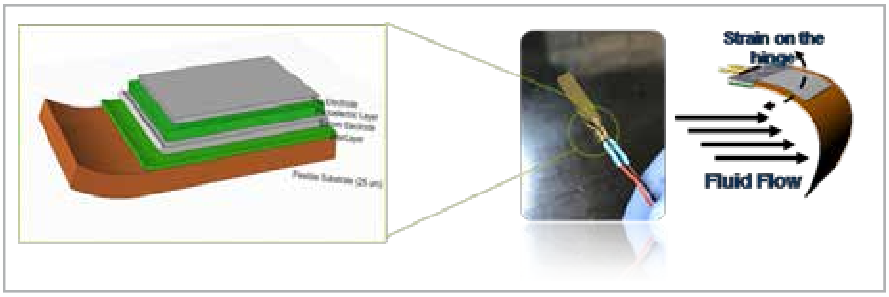


Fig. 7 - Transducer layout

Voltage generation occurs when these flags are deformed by the exhaust flow. The intensity of the voltage and the electric power increase as the deformation of the flag increases. The piezoelectric properties also affect the material properties, since the flag's stiffness increases with the quantity of current generated due to the conservation of energy.

The Strouhal correlation and others aerodynamics theories can be used to relate the body size and flow characteristics to the vortex shedding frequency:

$$Re_D = \frac{\rho U D}{\mu}$$

$$St = \frac{f_s D}{U} = 0.198\left(1 - \frac{19.7}{Re_D}\right)$$

where D is the characteristic dimension of the body, U the stream velocity, p the density, μ the dynamic viscosity, Re_D the Reynolds number, f_s the shedding frequency, St the Strouhal number. This makes it possible to estimate the shedding frequency and then design the flexible flag so that its natural frequencies match the vortices in order to reach the resonance condition and amplify the deflection of the flag. The assumption of a cylindrical bluff body can only be used as a starting point because the bluff body forms one single body with the flag and therefore cannot be considered isolated: simulation is crucial here to estimate the real vortex shedding frequencies.

The turbulent structures are strongly dependent on the flag's presence and on its time-varying deformation. In conclusion, this problem must be addressed with a two-way FSI approach because the physical coupling between the fluid behavior and the flag deformation is very strong and they affect each other considerably.

Numerical coupling of ANSYS solvers

ANSYS Workbench allows you to couple computational fluid dynamics (CFD) solvers (CFX or Fluent) and finite element method (FEM) solvers (Mechanical). This coupling is easily achieved using the system coupling tool that manages the interpolation mapping procedures and the numerical schemes needed to perform an FSI analysis.

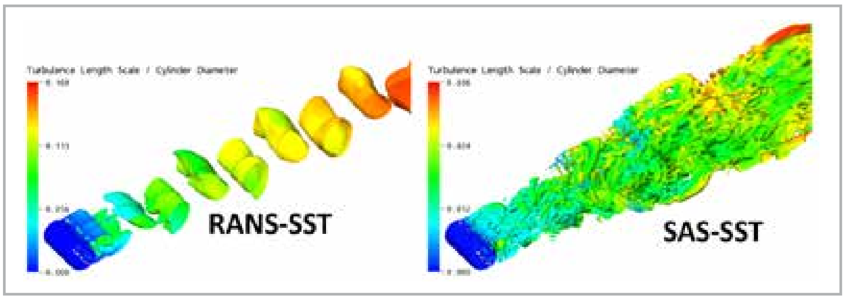


Fig. 10 - Example of RANS vs SAS-SST models

The implicit numerical scheme was chosen to solve this problem. (Fig. 6 illustrates how it works.)

Basically, with the implicit scheme, several coupling loops are executed between each time-step. In this way, the numerical stability increments compared to the explicit scheme and allows you to increase the time-step size. The structural and fluid-dynamics solvers exchange data several times until force/displacement convergence is reached. Within each coupling loop, field loops are executed inside the solvers to calculate the single physics.

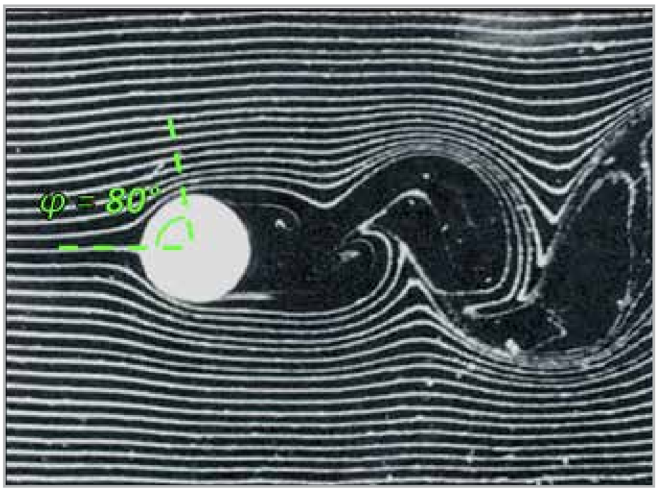


Fig. 8 - Boundary layer separation over a bluff body

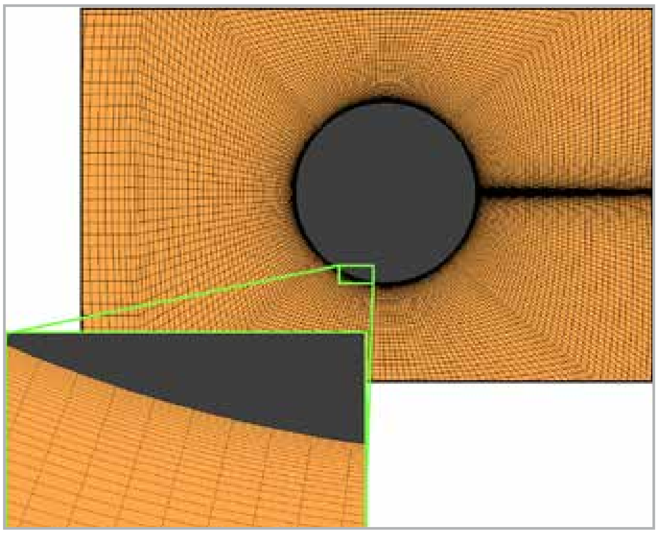


Fig. 9 - Mesh inflation refinement for Low-Reynolds wall treatment

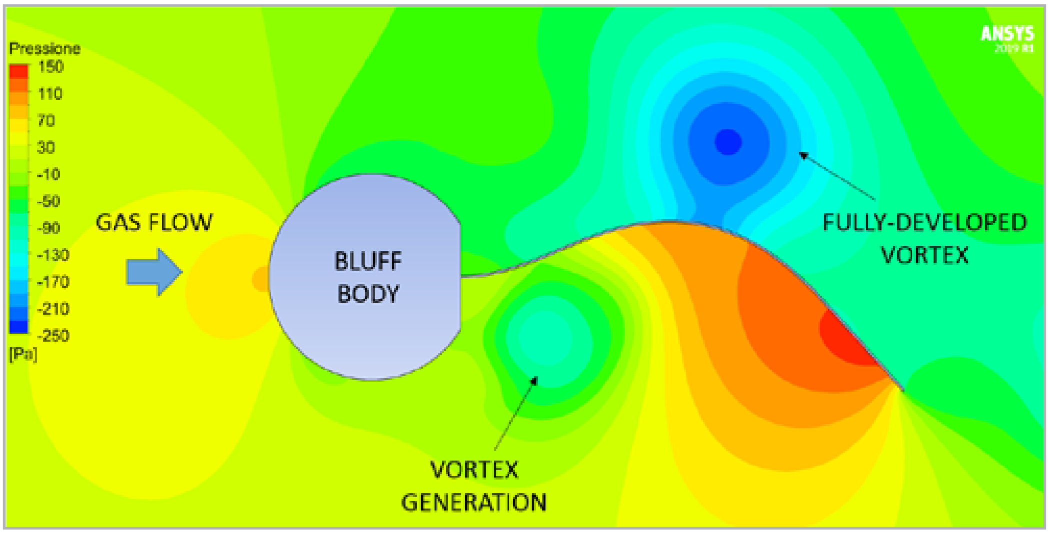


Fig. 11 - Example result of an FSI simulation of a flexible piezo-electric flag

Flexible flag and piezo-electric properties

Several piezo-electric materials are being investigated. So far, the candidate that best satisfies the requirements is Aluminum Nitride (AIN):

- Good piezoelectric coefficients (4-5 pm/V)
- High temperature performance (up to 1150 °C)
- Biocompatibility
- Grows on different flexible substrates (polyimide, kapton tape, PEN)

The structural model of the flexible flag is assembled in ANSYS Mechanical. Shell elements are chosen to model the device. ANSYS Mechanical makes it possible to build several layers with different material properties to replicate the material deposition process.

Piezo-electric properties are assigned to the materials to obtain the actual electric power generated by the fluid flow as the output of the analyses. These properties also affect the stiffness matrix assembled by the solver: the stiffness increases with the electric power generated, introducing a further non-linearity in the FEM model. By coupling this model with the fluid flow calculation, the solvers reach the convergence iteratively, time-step per time-step, taking into account of all the physics involved.

Fluid domain, turbulence and wall treatment

Since FSI simulations have high computational costs due to the very large number of iterations required, the fluid domain is simplified to two dimensions in the first phase of analyses. Focusing on the device, one can observe that large turbulence structures (integral length) develop mainly around the bluff body; the axes of these vortices are directed parallel to the bluff body itself and to the piezo-electric flag.

The turbulence model chosen was k-ω SST because Low-Reynolds wall treatment ($y+ < 2$) must be used to faithfully estimate the boundary layer separation around the profiles. For this reason, a strong mesh refinement was inserted close to the walls (see Fig. 9). Fig. 8 and Fig. 4 explain how the separation positions considerably affect the generation of the vortex, depending on the flow regime.

The second phase of FSI simulations will make use of 3D models to enable the secondary effects of the turbulence structures to be captured, too. In this phase, the RANS approach will be discarded: large eddy simulation (LES)-oriented turbulence models (LES, detached eddy simulation -DES-, and scale-adaptive simulation -SAS- etc…) will be used to conduct a detailed analysis of the behavior of the large eddies.

Simulation output for the project workflow

The results of the FSI simulations can be used to compare and design different system configurations and geometrical dimensions, once the parameters have been defined. Considering the entire range of gas velocity [0 – 20 m/s], the main outputs that can be extrapolated from the analyses are:

- Mode shapes
- Stress/strain in the flag
- Vortex shedding frequencies
- Vortex intensity for different bluff bodies
- Vortex evolution and pressure load distribution along the flag
- Voltage and electric power
- …

The next steps for the FSI simulations in the REM Project

- Continuous development of a methodology for the CFD and FEM numerics (algorithms, mechanical models, piezo-electric models)
- Detailed development of the moving mesh algorithms for the Eulerian domain (mesh deformation, mesh overset, …)
- Create a 3D domain to take into account the secondary effects of the turbulence
- Apply advanced LES-oriented approaches (DES model, SAS model…)
- Conduct parametric studies
- Execute specific runs to train the reduced-order models (ROMs).

For more information
Alessandro Cinciripini - EnginSoft
a.cinciripini@enginsoft.com

Using design for optimization to increase competitiveness

Optimizing production work station ergonomics makes a significant contribution

By Antonio Parona
EnginSoft

In today's globally competitive, high-speed markets, companies that not only want to maintain their market position, but increase it against their competition, have to reassess their business models. This is particularly important in light of the difficult balance between supply and demand, the need to modernize entrepreneurial environments and, more generally, the current market conditions. This is where the nexus between technological innovation, the ability to adapt to changing market conditions, and competitiveness is as crucial as it is obvious. Moreover, technological innovation and the ability to adapt are both key to ensuring or maintaining competitiveness.

There are numerous distinct aspects to innovate and re-design in striving to achieve this objective, but they all concern quality in its broadest sense: namely externally perceived quality by customers and competitors, and internally perceived quality, which is what most directly affects the company's internal organization. The production process is strategic in satisfying both perspectives, and designing (or re-designing) it for optimization becomes the fulcrum for innovative decision-making.

The design of the production process for optimization implies defining the parameters and adopting the strategies that will enable the company to achieve the highest quality standards with the available resources and capacity. More precisely: the more accurate the design of the production activities, the more production costs, production capacity and time to market will be impacted.

Hence, the design (or re-design) requires a precise analysis of both the completely and/or partially automated activities of the production processes, as well as any interactions with it by human operators. Most particularly, it is this last aspect that is

often neglected or poorly evaluated which results in the generation of forecasts and models that can differ vastly from reality. To be valid, the analysis of the manual activities must determine the correct timings and methods for each individual activity in the production flow as precisely as possible.

This makes it necessary to map the entire workflow in advance, depending on whether it is being planned or re-designed, and to then classify its general characteristics and functions in order to identify any possible critical points. The final objective of the analysis is to create as linear a flow as possible to eliminate any "downtime" and wastage that may occur either in relation to the processing of materials, or to its handling (such as excessive handling). In fact, these inefficiencies represent a significant potential cause of increased industrial costs with an obvious increase in the final production costs. Furthermore, these profiles are often found to be linked to other inefficiencies, such as the use of incorrect methods, inadequate work stations and tools, or errors in the order of the execution of the various production activities. All of these aspects must be innovated for an organization to recover profitability.

EnginSoft Turkey achieves BS EN 9100:2018 certification for Aerospace Quality Management Systems (AQMS)

Dr. Şadi Kopuz, GM of EnginSoft Turkey announced in June that the BSI has certified EnginSoft Turkey for the provision of computer-aided engineering for the design, development, analysis and engineering of aerostructures, land vehicles, marine and space technologies. The AS 9100 (BS EN 9100) is the single common quality management standard for the aerospace industry.

He said, "This attests to our commitment to quality assurance for our customers through continual improvement and increased efficiency."

This preliminary analysis should be completed with a detailed definition of all the activities in the entire flow at a global level, and of the individual activities and operations at a local level, all of which are also involved in determining the performance standards necessary for the continuous and systematic control of the correct and efficient execution of the production processes. The flow must therefore be designed to safeguard its constant execution over time, and to ensure that the pre-established metrics for measurement are easily verifiable. It is equally important to define the tasks that are fluidly linked to each other.

In defining the detail of the individual activities, the various factors that constitute the different work stations must also be studied, such as: the visibility and accessibility of the workers performing the different tasks, the usability of the tools, and the handling of loads, which implies an assessment of the weights and the possible need for specific equipment. Another factor to consider are the operator's movements in performing the task, the distance and frequency of which must be evaluated to possibly reduce them. It is also necessary to ensure that the environmental conditions are appropriate for the performance of the work, as defined by law, and that they respect ergonomic principles, which are also subject to regulatory constraints. Even in the absence of precise regulatory obligations, this latter aspect intrinsically contributes to optimized work stations by ensuring limited psychophysical fatigue, one of the main causes of increased processing errors and work times.
The use of dedicated software, such as ViveLab Ergo, is crucial when designing or re-designing work stations to optimize the production process. This solution allows the work environments, the operators, and the interactions between them to be recreated, ensuring a detailed verification of the aspects mentioned so far, and eliminating the need to physically reproduce the different work environments to test the effectiveness of the potential solutions. The software's computing capabilities allow the contextual examination of a plurality of alternative solutions, reducing design and optimization times, decreasing the subjectivity of the analysis, and increasing the accuracy of the results.

It also allows one to instantly assess whether the solutions being tested meet the standards for ergonomic conditions and highlights risk factors that would otherwise require specific assessment, potentially extending the design period and compelling analysts to review the projects.

Finally, this tool also functions as an additional method to monitor the correct execution of activities, which is essential to ensure adherence to production times and costs. This makes it unnecessary to identify additional tools for work measurement, making it another useful aid for optimization.

For more information
Antonio Parona - EnginSoft
a.parona@enginsoft.com

The brave new world of virtual commissioning

Current market demands require new approaches to assure business imperatives are achieved

By Giovanni Borzi
EnginSoft

Industrial automation customers, like any other customer, want higher quality products sooner at a convenient price, along with great after-sales service and support. Fortunately, virtual commissioning enables quality, time-to-market, and cost efficiency all to be greatly improved.

Virtual commissioning is a new approach for the design of automated machines that integrates all, or most of, the machine system elements, such as mechanics, electrics, sensors, actuators, robots, and the automation software from the very beginning of the design phase.

This enables companies to ensure the overall efficiency and quality of their machine's hardware and software, minimize the risk of errors, and dramatically reduce commissioning time. The following article explains how this new approach to the design of automated machines can enable industrial automation OEMs and system integrators to achieve these business imperatives with the assistance of industrialPhysics simulation software.

Industrial automation customers are no different from any other customer: they want their products delivered sooner, with a higher quality, at a convenient price. They also want to be the first to receive new products with great after-sales service and support. Fortunately for automated machines OEMs and system integrators, quality, time-to-market and, ultimately, costs are all objectives that can be greatly improved by adopting a virtual commissioning approach.

Whilst traditional commissioning involves a late (and oftentimes troublesome) “marriage” of mechanics, sensors, actuators and automation software, virtual commissioning is a new approach to machine design that develops the hardware and software in an integrated way from the very beginning of the design phase.
Since ever-tighter development schedules increase project risks, potential issues need to be identified and addressed effectively from the outset. Identifying an issue at a later stage will postpone delivery, potentially exposing the provider to penalties and, if the issue affects critical requirements such as machine performance, ruining the project and the provider's reputation with the customer.

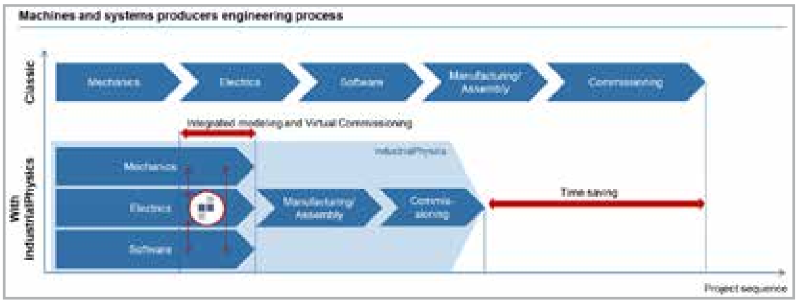


Fig 1 - Virtual Commissioning process with industrialPhysics

Errors must be ruled out from the start so that "right first time" designs can be achieved.

Customers require optimized design and the development processes need to enable true design optimization from the early concept stages, as opposed to late stage design improvements. Furthermore, innovative simulation-based design processes offer a tested and robust route for development teams to achieve new efficiency gains.

Virtual commissioning with industrialPhysics

Virtual commissioning can be defined as a design approach that integrates all, or most of, the machine system elements, such as mechanics, electrics, sensors, actuators, robots, and automation software.

industrialPhysics, a simulation software developed by machineering GmbH & Co. KG, is a powerful virtual commissioning environment.

industrialPhysics
simulate | automate | produce

Fig. 2 - industrialPhysics by machineering GmbH & Co. KG

It enables the simulation model to be easily obtained by importing the Mechanical CAD data (the geometry, and the material, mass, inertia information etc.) and then enriching it with simulation-specific information. The link established with the MCAD software is bi-directional: simulation information can either be saved back into the CAD model files, or into a separate file linked to the CAD model structure, as required. These features allow engineers to effectively integrate their industrialPhysics simulation models with PLM/PDM solutions and processes. Moreover, the industrialPhysics engine uses a differential synchronization technology that will only synchronize those parts of the CAD file that have been updated or changed: these feature, supported by an advanced caching engine, allow the designer to switch seamlessly between the MCAD and the virtual commissioning design environments.

Included among the various simulation information with which industrialPhysics can enrich the CAD model for designers are static, dynamic or kinematic objects. Moving parts such as products, conveyors, drives and actuators can be defined at the required level of detail. Mass, inertia, friction and collision can be accurately modeled. Sensors can be modeled as well, from simple contact sensors, light sensors and barriers, to barcode scanners and advanced pattern recognition. Once the simulation is started, rigid body dynamics simulation takes over; part flow and the machine's behavior can also be accurately simulated.

The automation software can be modeled using several approaches. All the active components of the model expose their I/Os to the software environment. A first approach is therefore to program the automation directly into industrialPhysics, using its own scripting language and the internal PLC emulation technology. This makes it possible to easily create and program machine simulation

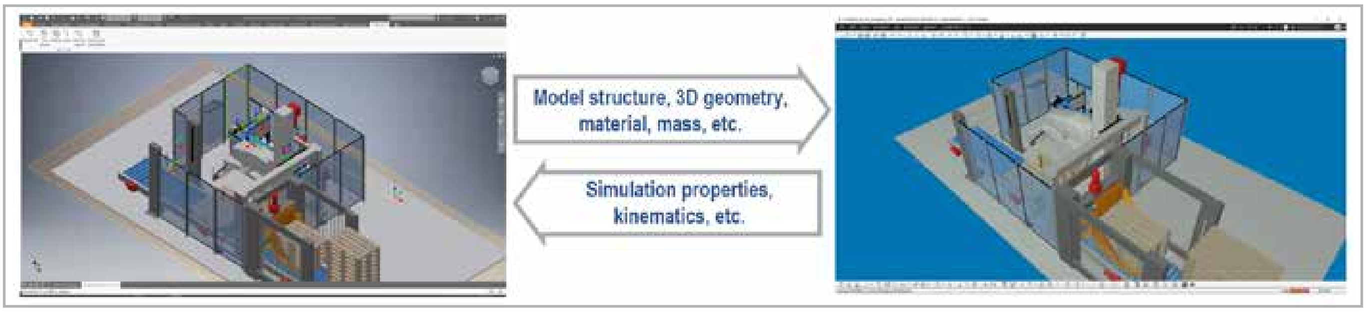


Fig. 3 - Fast bi-directional link with MCAD

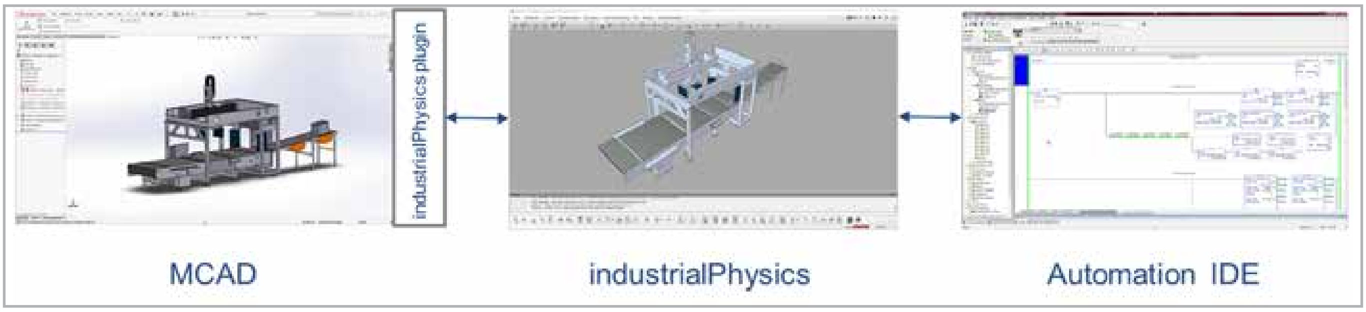


Fig. 4 - Automation developers can program in their current development IDE and test the software on the industrialPhysics simulation model

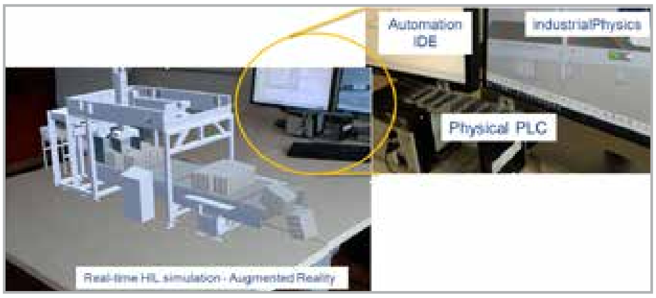


Fig. 5 - Real-time HIL machine simulation with industrialPhysics

models. A second approach is based on an I/O image that can be generated with ease by industrialPhysics and then used via a TCP/IP connection. The PLC software can be programmed by the automation software developers in the same way that they would for the real machine and using their existing development tools. The automation software connects to the virtual PLCs emulated in industrialPhysics.

A third approach exploits the hardware in the loop (HIL) capabilities of industrialPhysics. It is, in fact, possible to connect the simulation model to one or more physical programmable logic controllers (PLCs) using the same I/O values mentioned before. Fiedbus can either be emulated by industrialPhysics, or by an external device called Field Box 1, also developed by machineering GmbH & Co. KG. One of the main advantages of the Field Box technology is that the simulation PC's central processing units (CPUs) do not need to handle the fieldbus traffic as this is done in the Field Box, which increases performance.

Additionally, it is possible to connect PLCs from different vendors and using different protocols: in this way it is possible to emulate multiple devices on the Field Box, which generates a simulation that runs virtually as it would when deployed on the real machine counterpart.

Furthermore, industrialPhysics offers a comprehensive Robots library that allows machine designers to integrate prebuilt models of robots from several vendors, such as Fanuc, KUKA, Stäubli, Yaskawa etc. on the fly. The robot in the MCAD system can be enriched with inverse kinematics and path planning objects that convert the 3D geometry of the robot into a dynamic object that is managed by industrialPhysics.

The benefits of virtual commissioning: results of a field study

Virtual commissioning allows the creation of comprehensive models of a real system to enable better and faster development and testing. Using virtual commissioning, the physical behavior of the machine under development can be simulated long before any component of the system has been procured or manufactured. Real-time models can be used to validate new designs at an early stage, optimizing the system and guaranteeing the customer's specifications (e.g. throughput) will be met. The main benefits can be summarized as follows:

- risk reduction for complex projects;
- the possibility to evaluate a higher number of potential design solutions;
- machine sequence optimization;
- realistic estimation of throughput and overall equipment effectiveness (OEE);
- realistic production process visualization;
- simplified commissioning;
- transparent change management;
- increased software quality.

How can these benefits be measured? A field study titled "Head start with simulation" was run on real commissioning objects and by real development teams, with and without the use of industrialPhysics, and the key performance indicators (KPIs) were calculated. (Details of this study are available on request. Please see the contact details provided at the end of the article.)

The results demonstrated that virtual commissioning enables designers to:

- shorten commissioning time up to 75%;
- shorten entire project time up to 15%;
- enhance software quality by more than 40%;
- reduce costs up to 30%.

It can therefore be concluded that simulation tools such as industrialPhysics can be used today to perform comprehensive virtual commissioning, from testing early-stage simple logical

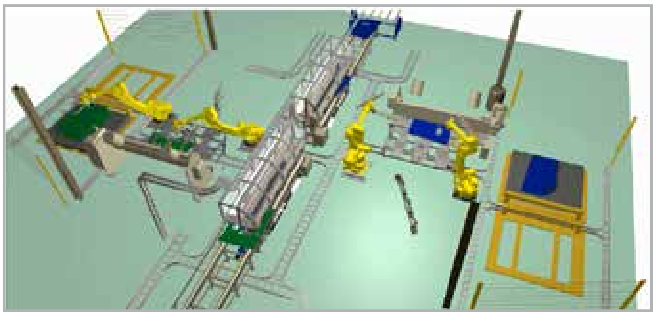


Fig. 6 - Simulation of robotic cells

sequences to complex and critical scenarios, to ensure the overall efficiency and quality of the machine's hardware and software, minimize the risk of errors, and dramatically reduce commissioning time.

Images courtesy of:
machineering GmbH & Co. KG

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

Simulating chemical processes to optimize plant and pipeline design with Xpsim

Demonstration of the transportation of CO_2 and LPG via a pipeline

By Luigi Raimondi
PSS

This article describes the use of Xpsim, a general chemical process simulator, initially developed for the simulation of chemical processes, and used for the design of chemical plants mainly in the field of oil&gas plant engineering. The software's libraries allow chemical engineers to study the chemical processes and then design production plants geared for the synthesis of the chemical components the plant will handle. A case study of a dynamic simulation of the transportation of LP gas via a pipeline is provided.

Xpsim (eXtended Process SIMulation) is a general chemical process simulator, initially developed for the simulation of chemical processes, to be used for the design of chemical plants mainly in the field of oil&gas plant engineering. The software can be used to simulate both steady state and dynamic problems. The software architecture is based on a number of libraries of the basic chemical-physical data of the most common chemical compounds found in oils and gases, and of methods for the calculation of their properties.

The library of chemical components contains the data of water, natural gases (nitrogen, oxygen, hydrogen, carbon dioxide, etc.), hydrocarbons (saturated, such as methane, ethane, propane, etc., unsaturated, such as ethylene, propylene, acetylene, etc. and aromatic, such as benzene, toluene, xylenes, etc.), alcohols (methanol, ethanol, etc.), aldehydes, ketones, esters, and so on.

The library of methods is based on up-to-date thermodynamic correlations, and can generate the values of the thermo-physical properties of pure components and of their mixtures, such as enthalpy, entropy, density, viscosity, thermal conductivity and fugacity for vapor, liquid and solid phases. The property of fugacity, which may sound strange to mechanical engineers, is of fundamental importance in the field of chemical engineering because it allows the calculation of the separation of the vapor and liquid phases.

Using these two cornerstones, a chemical engineer or an industrial chemist can study the chemical processes and then

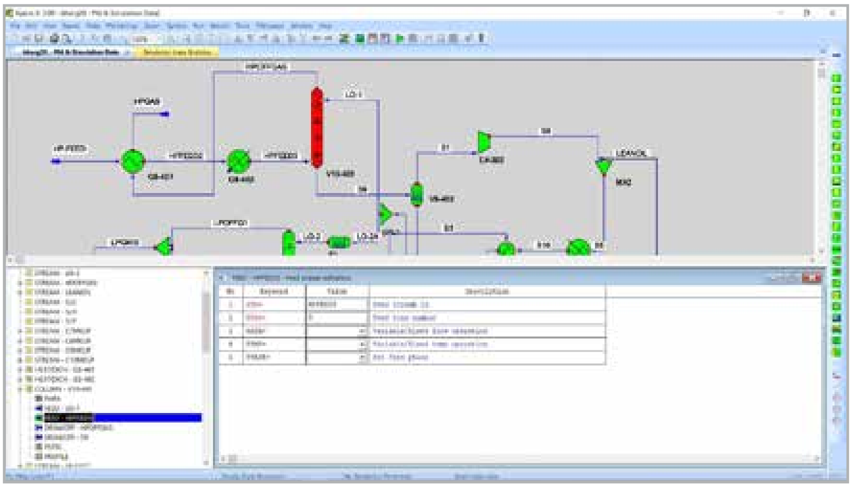


Fig. 1 – A PFD as depicted by the Xpsim user interface

design production plants geared for the synthesis of the chemical components fundamental to the development of the synthetic materials used in the production of the goods we use everywhere today (fuels, plastics, pharmaceuticals, etc.).

The first step in the design of a chemical plant is the definition of the process flow diagram (PFD) as the starting point for obtaining the heat and material balance (HMB) of the plant. Therefore, the process engineer must set up a steady-state simulation of the plant and define the chemical compositions of the inlet mass flows and the basic operations (unit operations or UOP) required to obtain the desired products.

These operations may be mixers, splitters, vessels, component separators, distillation columns, compressors, expanders, pumps, heat exchangers, control valves and pressure safety valves. Some of these operations correspond to real items and some to logical ones. The connections between the operations can be simple pipes or pipelines for the long-distance transport of oil and gas.

All these steps are performed using a process simulator: the connection diagram is created using a Windows interface that allows users to define the required input data. An example of a ‘not simple’ chemical plant is shown in Fig. 1.

After the user has completed the input data, he can run a simulation and analyze the results. Complete results may range from a few to several hundred pages. The user can preview them in sequence or jump from the output index to the desired page. Over the years, Xpsim has been extended to incorporate a multiphase pipeline for transporting oil and gas mixtures as a single operation. This feature was driven by the need to simulate oil and gas production in the offshore reservoir and the transport of well products to onshore facilities.

The thermodynamic method library includes state- of-the-art cubic equations of state, such as the Peng-Robinson (PR) and the Soave-Redlich-Kwong (SRK), or newer models such as the Cubic-plus-Association (CPA), the PC-SAFT, and others. For the calculation of enthalpy and entropy, the cubic equations are also available with the Lee-Kesler (LK) and the Benedict-Webb-Rubin (BWRS) models. Many models are available to calculate vapor and liquid densities, and to evaluate transport properties (viscosity and thermal conductivity), which are of fundamental importance when designing the transport of vapor and liquids across long distances.

When transporting oil and gas, the liquid and vapor phases accumulate in different quantities along the pipeline as a result of the different inclinations imposed on the pipeline by the profile of the ground, and due to the different densities and viscosities of oil and gas. Therefore, when the pipeline is horizontal, the level of the liquid increases due to its higher viscosity, while the gas will increase its speed due to the increased space available in the lower section of the pipe. In the case of an upward inclination, the level of the liquid will increase to the point at which it begins to flow backwards, meaning that the two phases will flow in counter-current mode. This creates a situation where the gas and the liquid flowing through a long-distance line establish a kind of a quasi-periodic dynamic flow in which the liquid-slugs are pushed ahead by the increased gas pressure caused by the accumulation of liquid in the lower parts of the pipeline.

When a dynamic simulation is set up, the description of the problem is more related to the real plant equipment: valves must be defined by giving the flow coefficient (Cv) value; pipes and pipelines are characterized by their diameter, length and elevation, compressors by their operating curves; and instruments and control circuits can be added. In addition, the user can provide data to initialize the simulation at time 0, define the final calculation time and events such as opening and closing a valve tosimulate emergency

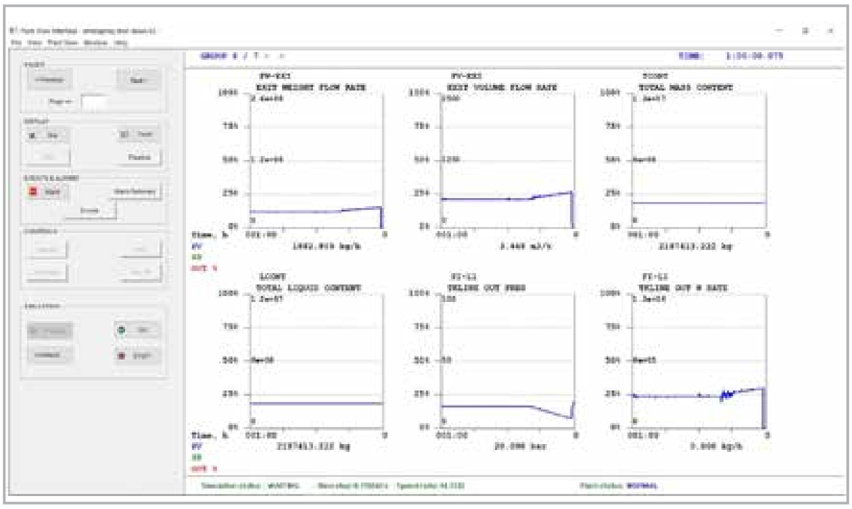


Fig. 2 –dynamic simulation in Xpsim – user interface view and control

operations or normal changes during normal operations, start-up and shut down. When a dynamic simulation is started, a second new window is created, as shown in Fig. 2.

This interface offers a view similar to the one displayed by digital control systems (DCS) on chemical plants, or by supervisory control and data acquisition (SCADA) systems on pipeline networks. The user can freeze the simulation at any time and browse through what is going on by clicking the instrument values.

Xpsim steady-state simulation has already been used in the design of many pipelines, for example the blue stream used to export Russian natural gas to Turkey across the Black Sea at a depth of over 2000 meters.

An example of the capabilities of Xpsim is taken from a presentation prepared for an event held in 2014 titled "CO_2 Transportation with Pipelines - Model Analysis for Steady, Dynamic and Relief Simulation". The transportation of a mixture of captured carbon dioxide (rich in CO_2 with small amounts of methane (molar composition: CO_2 94%, CH4 4%, N2 2%)) was simulated. The profile of the pipeline is illustrated in Fig. 3. Fig. 4 shows the pressure, temperature and hold-up profiles along the 525km pipeline.

The results of a simulation of the situation in summer case identified that the fluid partially vaporizes in the final segment of the pipeline along with a sharp drop in the pressure due to the two-phase flow. The final liquid fraction becomes 0.25.

In Fig. 4 we can see that Xpsim correctly predicted that when the fluid passes through the region of the critical point (31.01°C - 73.8 bar for pure carbon dioxide), the liquid hold-up jumps from 0 to 1. Subsequently, the liquid begins to evaporate, the two-phase flow establishes and there is a sharp drop in the pressure.

As an example of dynamic calculation, let's consider a 50km-long pipeline (diameter 0.320m) transporting 400 m3/h of liquefied petroleum gas (LPG) (molar percentage composition: ethane 2, propane 30, n-butane 67, n-pentane 1).

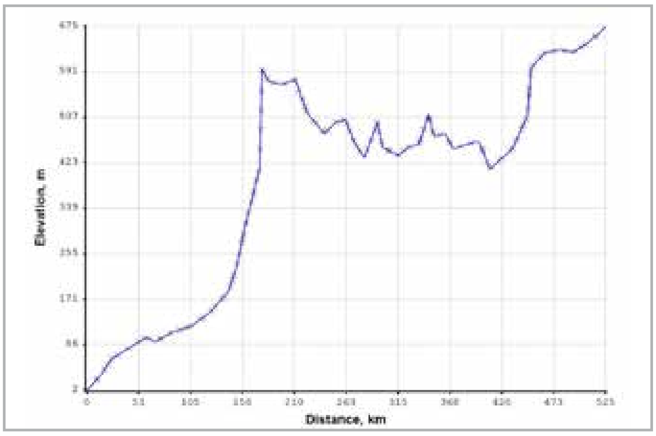


Fig. 3 – Pipeline elevation (x) vs distance

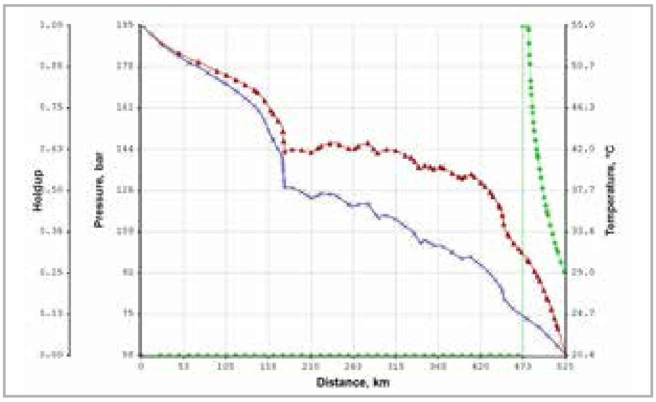


Fig. 4 – Pressure (x), temperature (Δ) and liquid hold-up (◊) profile

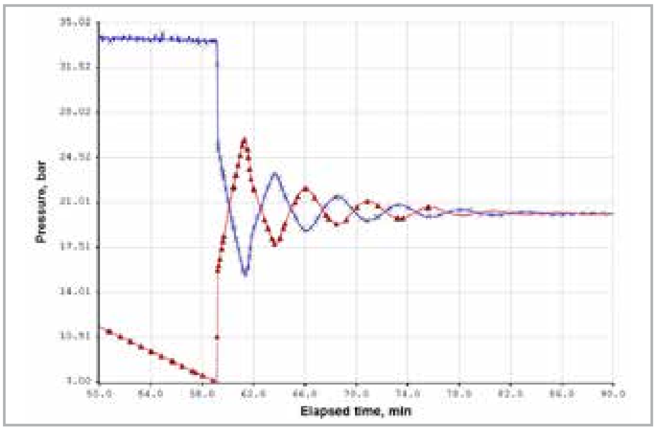


Fig. 5 – Inlet (x) and outlet pressures (Δ) vs time

The difference in upward elevation between the 10km and 20km points is 200m until it reverts to sea level at the 30km point. The inlet pressure is about 25 bar and the outlet pressure is maintained at 15 bar. Due to a fault in the receiving station, the outlet pressure begins to decrease until it finally reaches 7 bar, triggering an emergency event that closes the outlet and inlet control valves in 1 and 5 seconds respectively. This event would generate a fluid hammer phenomenon at the 60th minute of the simulation. Fig. 5 shows the inlet and outlet pressures over the 50- to 90-minute period: pressure waves travel along the pipeline until the effects of friction bring the fluid to rest.

At the upper point of the pipeline profile, the pressure may be below the bubble pressure when a certain amount of vapour is generated. In fluid hammer simulations, the speed of sound is an important value, and Xpsim calculates thermodynamically consistent values. Compared to water, LPG is more compressible and its speed of sound is about 500 m/s. These values become lower, reaching 70 m/s where the vapor and liquid phases coexist.

Other examples of using Xpsim to solve important engineering problems are available on the web at: www.xpsimworld.com.

For more information
Livio Furlan - EnginSoft
l.furlan@enginsoft.com

How is the wall thickness of real bends for piping systems addressed in PASS/START-PROF software?

By Alexey Matveev
PSRE

ASME Code oversight creates problems for piping stress engineers

Piping stress engineers often have to deal with an issue that can cause them to underestimate the pump, nozzle, and support loads and stresses that are calculated by the ASME B31 codes!
The ASME B16.9 and all the ASME B31 codes don't regulate the bend, tee and reducer wall thickness (WT); only the pipe WT is regulated. As a result, some piping engineers think that the elbows and other fittings have the same or similar WT as the matching pipe's.
But, in most cases, the real WT of the bend, tee, and reducer bodies is greater than the WT of the matching pipe with the same schedule.
For elbows, the real WT can be from 10% to 40% greater than that of the matching pipe, while bends must have greater WT to contain the same pressure as a connected straight pipe (paragraph 304.2.1 3d of ASME B31.3).

Manufacturers usually produce bends with greater WT than the matching pipe's, but the real bend's WT value can, as a rule, only be obtained by contacting the manufacturer, or by measuring the real bend thickness after delivery.

Piping designers usually know nothing about this, and, when using piping stress analysis software, piping stress engineers often use the pipe's WT for the elbows. For instance, not inserting a value in the "Fitting Thk" field (in CEASAR II) lets the software assume that the elbow has the same WT as the connected pipe element — but this is a serious mistake!

According to the ASME B31 and other ASME B31-based codes, the bend flexibility factor depends on the real bend's WT, and not on the matching pipe's WT.

The greater the bend's WT, the greater the bend's stiffness (k-factors) and the greater the loads will be on the rotating equipment, nozzles, supports and the expansion stresses in the piping system.

This problem quite often comes to light when Russian companies try to check designs for the Russian market that have been made according to the ASME B31 codes. When rechecking the stress analysis using the most popular software on the Russian market (and which is certified and accepted by authorities), namely PASS/START-PROF which uses the GOST codes, many errors are reported. The reason of this is that the WT of the elbows being lower than the minimum required to contain the pressure because it was usually left blank in CAESAR II, and, in the piping stress model, the software then assumes that the fitting thickness is equal to the connected pipe's WT.

Once the real WT of the elbow is entered and the model is recalculated, the nozzle loads and stresses become much greater than those calculated by CAESAR II or other software. This is because the elbow flexibility k-factors used during the analysis were incorrect.
But in reality, the manufactures cannot usually provide the real WT of the bend body because they just don't have this information.

PASS/START-PROF's team obtained different answers to the direct question: "What is the real WT throughout the whole bend body?":

Fig. 1 - Fittings database in START-PROF

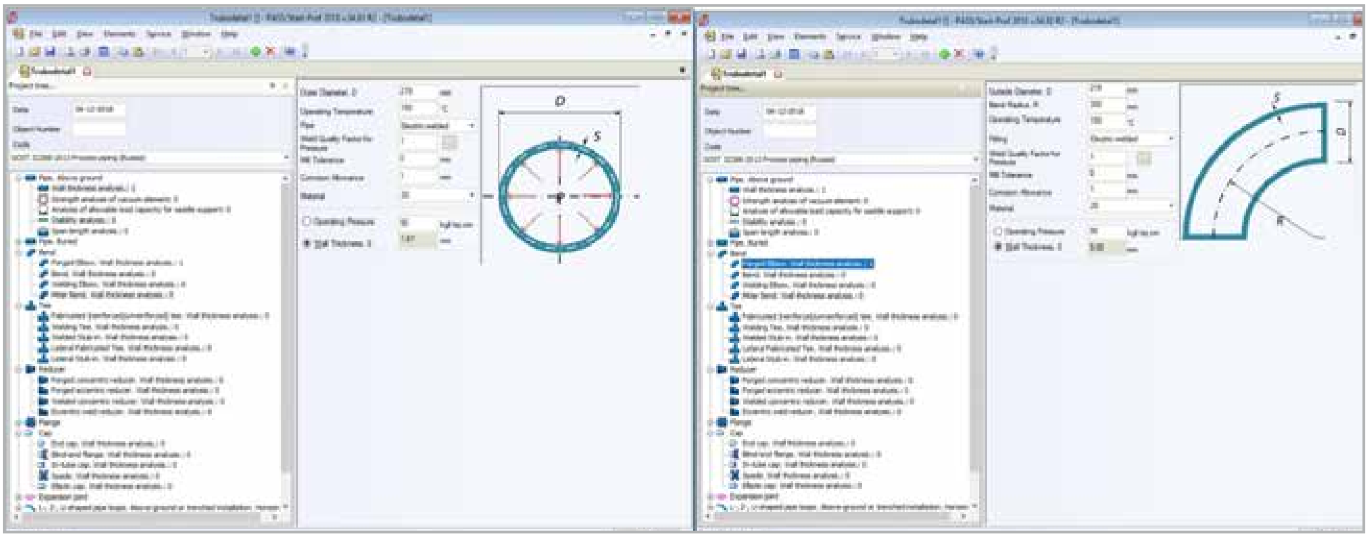


Fig. 2 - Calculated WT for pipe and bend

- Some manufacturers did not answer at all;
- Others said it was a trade secret (?!);
- Some manufacturers replied that the edge would be 100% consistent with the ASME Schedule, but that the thickness of the wall at the bend could even be 40% greater!

It is only possible to measure and determine the real WTs once the ordered fittings (bends, reducers and tees) have been received from the factory. This means that the stress analysis model has to be changed and that the nozzle loads become greater, which requires

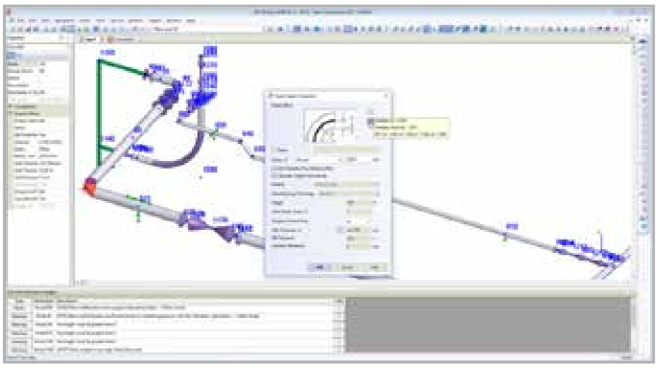


Fig. 3 - Defining elbow in START-PROF

the design to be changed to add more flexibility and to reduce the nozzle loads and, sometimes, the expansion stresses. And all of this has to be done after the contractor's design job has been formally "finished". Amazing!

In contrast to ASME, the GOST standards, which are completely different to ASME B16.9 for bends, tees and reducers, always provide the real body WT for each fitting. Manufacturers follow the standards. Every piping stress engineer, therefore, knows the real body WT of the bends and the other fittings, and specifies them in START-PROF when performing piping stress analyses. All these bend properties can also be taken from the fitting database (Fig.1).

All RD, GOST, and SNiP stress analysis codes (for power, process, oil & gas main pipelines, etc.) provide the detailed WT calculation procedures for all fittings, including bends, tees, and reducers. In Fig.2 you can see that the calculated bend WT is always greater than the pipe WT for the same pressure.

The real bend body's WT should be used for piping stress analysis instead of simply matching the pipe wall's thickness. To solve this problem, the PASS team added a special feature in PASS/START-PROF software that allows the calculation of the approximate "real" bend WT on the fly, according to the ASME B31.3, 304.2.1, and similar requirements in other ASME B31 and EN 13480 codes. By just clicking on the "C" button near the "Wall Thickness" field (Fig.3), the real bend WT will be calculated according to the code requirements.

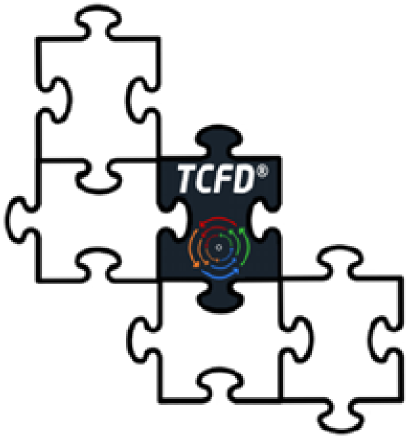
Conclusion

1. Bend, tee and reducer WT should be regulated by the ASME B31 codes and provided in the ASME B16.9 code. Manufacturers should produce bends with a body WT in accordance with the code requirements.
2. Until this first problem is resolved, manufacturers should provide the bend WT in their catalogs to allow designers and piping stress engineers to use the real WT in their pipe stress models in order to enable them to obtain accurate nozzle loads and expansion stresses.
3. A special remark should be included in the ASME B31 codes that explains how to calculate the flexibility k-factors for elbows where the real body WT is unknown.
4. If the elbow WT is unknown, piping stress engineers should use the WT calculated by the ASME B31 code equations for the bend, or use the pipe WT multiplied by a factor of 1.4. This will lead to the generation of a more conservative design so that, once the real bend WT becomes available (or can be measured), the changes required in the piping design will not be as critical as they currently are.

For more information
Livio Furlan
l.furlan@enginsoft.com

Benchmark validation of a complex CFD simulation using TCFD

Simulation results agree well with measurement data



By Lubos Pirkl
CFD Support

The following article is based on the public part of the report on the benchmark validation of a CFD simulation of a centrifugal fan using TCFD®. This project was undertaken jointly by ZVVZ MACHINERY, a.s. in collaboration with CFDSUPPORT. The aim of the benchmark validation was to evaluate the TCFD® computational fluid dynamics (CFD) simulation software and compare its results with the measurement data. The secondary aim of this project was to investigate the difference between steady-state and transient simulation results.

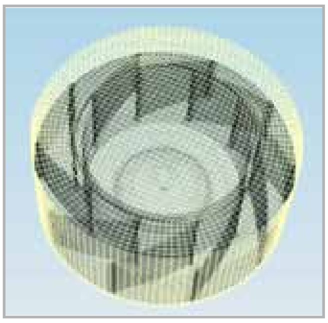
Mesh stats	Impeller	Spiral
points	2542103	3074101
faces	6881199	7920747
internal faces	6541317	7271691
cells	2174548	2433864
faces per cell	6.172554	6.242106
hexahedra	1906820	2117826
prisms	68676	67025
tet wedges	842	607
tetrahedra	35	8
polyhedra	198175	248398

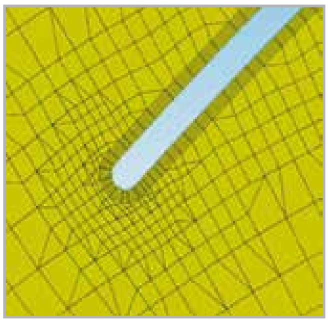
Benchmark Parameters

- Typical flow speed: 60 m/s
- Flow model: incompressible
- Rotation RPM: 1200
- Mesh size: 2.8M cells
- Impeller diameter: 1.0 m
- Medium: air
- Reference pressure: 1 atm
- Typical Pressure Ratio: 1.03
- Reference density: 1.2 kg/m^3
- Dynamic viscosity: 1.8 × 10^{-5} Pa·s
- CPU Time Steady: 6 core hours/point
- CPU Time Trans.: 20 core hours/point

Preprocessing

In this particular project, the mesh for the CFD simulation was created from the surface model via an automated process (alternatively, the external mesh can be loaded directly). The original CAD model of the centrifugal fan was in the STEP file format. Original STEP files are usually too complex for comprehensive CFD simulations, so certain preprocessing CAD work is generally required. While the original CAD model for this project was simplified and cleaned using Salome open-source software, any other standard CAD system can be used instead. The principle is always the same: the surface model has to be created; all the tiny, unimportant, and problematic model parts must be removed, and all the holes must be sealed up. This model is split reasonably into individual waterproof components. Then, the final simulation-ready model, the 3D surface in STL file format, has to be refined to a reasonable level. This preprocessing phase of the workflow is extremely important because it determines the simulation potential and limits the CFD results.





Mesh

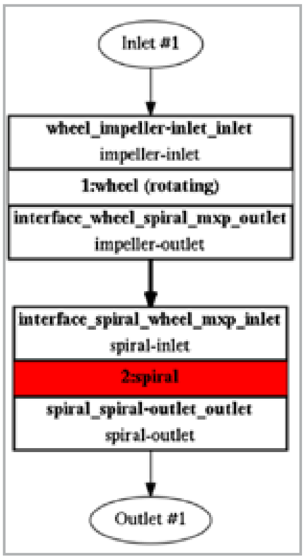
The computational mesh was created in an automated workflow using snappyHexMesh. A cylindrical mesh was used for the initial background mesh of the fan impeller. The mesh refinement levels can easily be changed to obtain a coarser or finer mesh to better handle the mesh size. Then, a boundary layer mesh of 5 inflation layers was created on the wall surfaces. Alternatively, an external mesh can be loaded.

TCFD® Case Setup

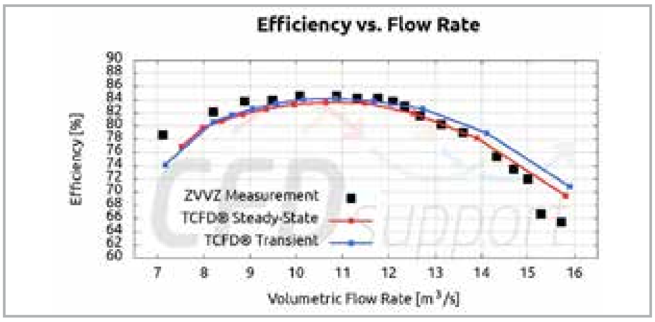
Basic TCFD® settings:

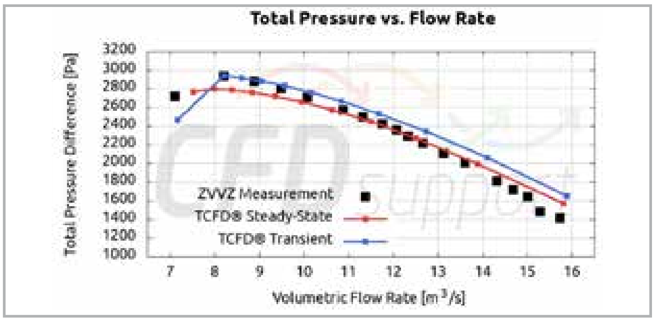
- Machine type: fan
- Both Steady-state and Transient calculation
- Incompressible fluid flow
- RANS turbulence modeling with k-ω SST model
- Number of components: 2
- Mesh size: 4.6M cells
- Inlet: Flow rate
- Outlet: Average static pressure
- Interface into Impeller: AMI
- Interface out of Impeller: Mixing Plane (10 planes)

The simulation was executed in the automated workflow in both steady-state and transient modes. The single speed-line of 11 points at 1200 RMP was simulated. TCFD® is able to write the results down at any time during the simulation. The convergence of any quantity is monitored during the simulation. When any simulation point converges sufficiently, the simulation can move onto the next simulation point.



Results comparison with the measurement data





Post-processing

TCFD® includes a built-in post-processing module which automatically evaluates the required quantities, such as efficiency, torque, flow rates, and forces and moments. All these quantities are evaluated throughout the simulation run, and all the important data is summarized in an HTML report, which can be updated anytime during the simulation for every run. Furthermore, visual postprocessing of the volume fields can be done with ParaView.

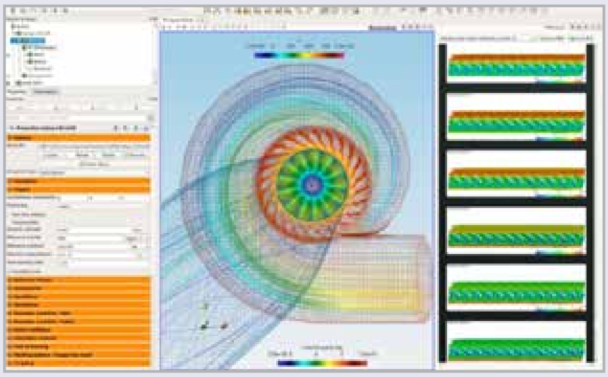
Conclusion

The complex CFD analysis of a centrifugal fan using TCFD® provides very good agreement with the measurement data, as has been demonstrated. Overall, the transient simulation's prediction is slightly closer to the measurement data than the steady-state simulation's.

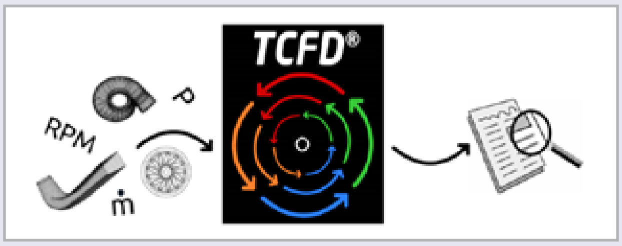
For more information
www.cfdsupport.com
info@cfdsupport.com

About TCFD®

CFD SUPPORT LTD creates a new generation of CFD simulation tools. TCFD® massively increases productivity in CFD simulations. It successfully merges the benefits of both open-source and commercial code: due to its open-source nature, TCFD® is perpetual for an unlimited number of users, jobs and cores, and it is further customizable; due to its commercial nature, TCFD® is professionally supported, well tested, industry-ready, robust, accurate and automated, offers a graphical user interface (GUI), documentation and much more.

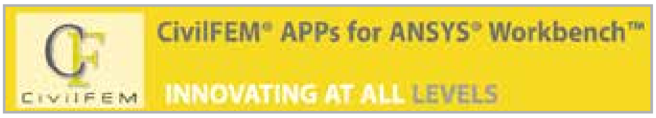


Since TCFD® is unlimited, it is well-suited to demanding simulations like optimization, transient or aeroacoustics. It scales CFD simulations to the available hardware resource and is fully automated and flexible. Another feature is that it allows the user to decide how deeply to dive into CFD, or whether to use it at all. TCFD® can be used as a black box (data in - data out), or as a sophisticated CFD code where all the options remain open at the same time. Originally designed for simulations of rotating machinery like pumps, fans, compressors, turbines, etc. TCFD® proved to be so effective that it was later extended with many other applications to cover an even a wider range of CFD uses. TCFD® performs well on the external aerodynamics of various objects. Its numerical solver is based on OpenFOAM®. The product is independent of other software but is fully compatible with standard OpenFOAM® and other software packages. TCFD® is fully automated and can run an entire workflow with a single command: from data input, the new case is written down, the mesh is created, the case is set up and simulated, the results are evaluated, and the results are written into a report. It operates in both GUI or batch mode. It has been developed purposely to fit into any existing CAE workflow. TCFD® has a modular (plugin) character: any part of it can be used within other applications and it has strong interfaces to cover a wide range of input and output data.



ANSYS announces new CivilFEM APPS for ANSYS Workbench

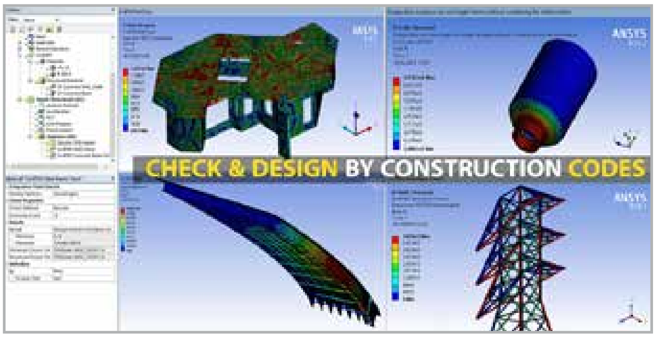
Enables construction check and design to the main international standards



ANSYS® Structural products are widely used by the most innovative companies in the construction industry to analyze the behavior of structures and assess their resistance to various scenarios.

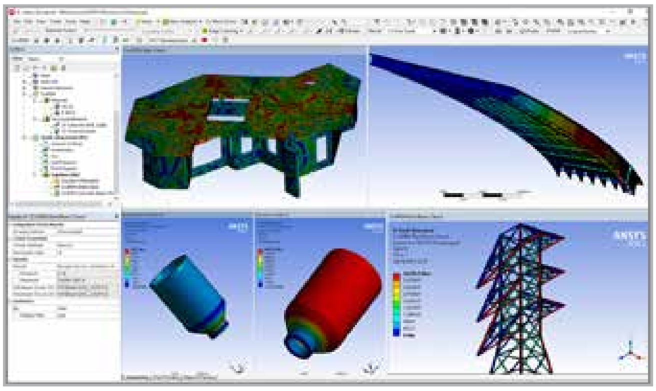
A much-demanded capability in the construction industry is check and design using the main international construction standards and, fortunately, ANSYS® users can now check and design their models using these standards with the help of the new CivilFEM® APPs for ANSYS® Workbench™.

The CivilFEM® APPs include check and design tools for Beams and Shells of both Steel and Reinforced Concrete for European



(EC2 and EC3) and American (AISC, LRFD, AISC ASD, ACI318 and ACI349) standards. The apps also incorporate a complete material library and section library that encompasses materials and sections from different standards. In addition, they allow the simple and intuitive generation of the load combinations, an essential task in the construction field for its later check and design.

This novelty comes from Ingeciber, a company specialized in computer aided engineering (CAE) technologies, consulting engineering services, CAE distribution, specialized training, and R&D projects. Ingeciber has more than 25 years of experience developing tools for the civil engineering, construction, advanced architecture and geotechnics fields in the ANSYS® MAPDL™ environment through CivilFEM for ANSYS®. As an ANSYS® Preferred Solution Partner, Ingeciber also develops and will continue developing other tools in this area, but this time as an add-on to ANSYS® WORKBENCH™



The new CivilFEM® APPs for ANSYS® Workbench™ are available at a very competitive price through ANSYS® resellers and on the official ANSYS® AppStore.

The following standards are currently available:

CivilFEM® Apps for ANSYS® Workbench™ Products:

- ***Eurocode Check & Design:***
  Eurocode 2 (EN 1992-1-1:2004/AC:2010)
  Eurocode 3 (EN 1993-1-1:2005).
- ***AISC/ACI Check & Design:***
  ACI 318-14
  AISC 14th Edition (LRFD and ASD).
- ***Nuclear ACI 349-13 Check & Design:***
  ACI 349-13 Nuclear Code.

In addition, the new On-demand feature means that new standards – or even a previous version of a standard – can be added on request.

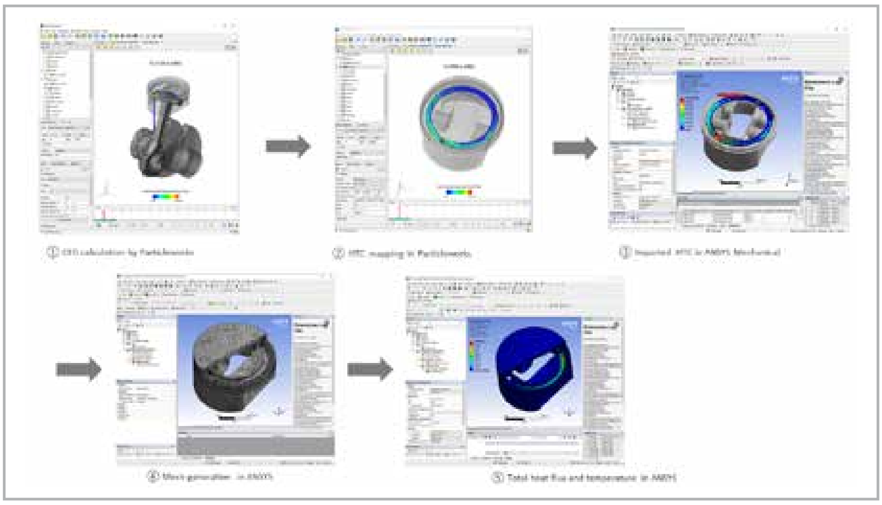
For more information, please visit
civilfem.com

Particleworks for ANSYS Workbench and ANSYS Mechanical now available

A direct interface enables the coupling of simulations between Particleworks and ANSYS

Particleworks, a liquid-flow simulation software based on the moving particle simulation (MPS) method, has been introduced in many industrial fields because of its mesh-free, easy-to-use operation and unique simulation capabilities required for fluid phenomena such as liquid drop, mixing, lubrication, spraying, sloshing and splashing. Its appeal to a broad range of users not only in the automotive industry, but also in the machinery, electric, material, food, chemical, biomedical, energy, and civil engineering industries all over the world has been growing over the past 10 years since its first release in 2009.

Today, demand for such complicated liquid flow simulation is increasing among niche users and a growing number of CAE engineers that work with structural, dynamic, thermal, electromagnetic, and/or general fluid flow analysis, who expend considerable effort to determine the influence of liquid flows to and from those physical behaviors. This is also true for users of ANSYS, the world-leading multiphysics simulation software.



HTC (Heat Transfer Coefficient) is produced by liquid flow calculated in Particleworks and mapped on the ANSYS model as a load condition for heat transfer analysis in ANSYS

To address the need to couple existing simulation areas with liquid flow simulation with large free-surface deformation, Prometech Software, Inc., the developer of Particleworks, recently released an interface product for ANSYS. Particleworks is now integrated into the ANSYS Workbench environment and can be combined with ANSYS' simulation capabilities for more realistic product design evaluations. Incorporating Particleworks into the ANSYS Workbench environment enables key features, such as calculating the deformation of a structure caused by fluid pressure, and the transfer of heat from a structure with the heat transfer coefficient.

Fig. 1 shows the simulation flow from the liquid flow simulation in Particleworks to the heat transfer simulation in ANSYS.

Particleworks can also be integrated with ANSYS Mechanical for calculating e-drives and the cooling of internal combustion engines. In this case, the information about heat transfer and cooling from the Particleworks simulation can be automatically and seamlessly transferred to ANSYS Mechanical to take into account the efficiency of the oil cooling.

ANSYS Mechanical will use the boundary conditions from Particleworks to accurately predict the temperature and stress of the e-drive or piston.

In addition, Particleworks can transfer the mechanical pressure distributions during a transient process, such as the pressure produced by the sloshing of fuel or any other liquid in a tank, or the forces produced by water, mud or snow on the bumper of car, to ANSYS Mechanical. In this case, the force history or time-dependent pressure distribution on the structure is mapped from the Particleworks' CFD simulation to the mechanical analysis to design and verify the structural resistance under dynamic loads.

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

Second edition of ESTECO Technology Days focused on MDO in aerospace

Multidisciplinary design optimization (MDO) is a rapidly growing market segment, as a result of the increasing global focus on lifetime cost containment across industry sectors. This made Aerospace MDO an obvious choice as the focus of the second edition of the ESTECO Technology Days, which took place on March 19-20, 2019 in the Ronald Reagan Presidential Library in Los Angeles, California. The two-day event featured a rich program of presentations by customers, with technology updates by ESTECO, and a workshop on the web-based collaboration framework, VOLTA.

The event was opened by Roel Van De Velde, Director of AS&D at ESTECO, who gave a brief introduction to ESTECO and explained the concepts and benefits of MDO using a recent success story from Embraer, a global aerospace company headquartered in Brazil. His presentation was followed by a historical overview of ESTECO, provided by the President of the company, Carlo Poloni, who illustrated the company's roots in aerospace by highlighting several case studies conducted during the EU-funded FRONTIER project in the mid-90's, which led to the birth of modeFRONTIER.

An overview of the Air Force Research Laboratory (AFRL)'s EXPEDITE program (Expanded MDO for Effectiveness-based Design Technologies) was provided by Clif Davies, EXPEDITE Program Manager, and Michael Mull, Conceptual Design Engineer at Lockheed Martin Aeronautics. This program intends to rapidly improve MDO capabilities in early conceptual design across several key areas, specifically Effectiveness Based Design, state-based modeling and path dependency, the transient operation of systems and subsystems, uncertainty quantification, high-performance computing, geographically distributed computing, and cost and reliability.

This presentation was followed by a modeFRONTIER case study provided by Ilan Paz, Systems Engineer at Crane Aerospace &

Electronics, which illustrated how the software is being used together with Matlab, Hardware-in-the-Loop and response surface modeling to derive real-time runway monitoring conditions.

The technology updates to be included in modeFRONTIER's latest 2019R1 release such as autonomous optimization algorithms, machine learning using H2O libraries, and simulation data management, were then presented byESTECO technology experts.

BAE Systems Ordnance Systems concluded the morning's proceedings, with Joe Bellotte, Rocket Development Manager, and Jonathan Jubb, Rocket Development Engineer, highlighting several applications of MDO in solid rocket motor design.

MDO took centre stage in the afternoon with presentations from Harsh Shah, Research Engineer at the National Institute for Aviation Research who presented on MDO methods to support virtual conceptual aircraft designs, and Ray Kolonay, Director of the Multidisciplinary Science and Technology Center at AFRL, who gave a US Air Force perspective on distributed collaborative MDO.

Thereafter, ESTECO Senior Application Engineer, Alex Duggan, shared several examples of cloud computing, and there were presentations on two current EU research programs: UTOPIAE and COMPOSELECTOR.

The first day's program concluded with ESTECO experts sharing the company's product development roadmap, and providing news on the VOLTA API, augmented analytics, the web workflow editor based on the business process model and notation (BPMN) standard, and product lifecycle management (PLM) integration.

The second day's workshop, which was well received, offered delegates the opportunity to explore VOLTA using a collaborative MDO exercise.

Poloni states, "Overall, the event was a great opportunity for industry and academia to share their experiences and voice their wishes for future technology developments. We are looking forward to the upcoming North American Users Meeting which will take place on Oct 29-30, 2019 in Detroit, Michigan, for further interaction with our customers and hope to see many people attending."

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

Innovative use of CFD for Turbomachinery demonstrated in Modena

EnginSoft event provided R&D engineers with ongoing education, networking opportunities and case studies

Industrial rotating machinery plays an important role today due to its wide application in the most disparate sectors, such as energy, oil&gas, aerospace, automotive, heating, ventilation, and air conditioning (HVAC) and consumer goods. In these markets worldwide, competition among producers makes innovation essential for competitive edge. As a consequence, designers involved in product development and fine-tuning have to balance the demands of reduced costs and time-to-market with those for product designs that feature the highest levels of efficiency, reliability and durability, while also preventing cavitation, harmful emissions and noise. The event held in Modena on April 16 and promoted by EnginSoft offered many of the engineers involved in the R&D of industrial rotating machinery the opportunity to see how these ambitious goals can easily be achieved using state-of-the-art simulation.
The presenters involved showed the best practices and the latest models being used by current market leaders to design and develop rotating machinery using reliable computational fluid dynamics (CFD) simulation. A specific session was dedicated to the development and simulation of rotary positive displacement machines such as gear pumps, gerotors, lobe and vane pumps, screw compressors, scroll compressors and expanders. The complex geometry of these machines derives from the variable volumes of flow in their work chambers. Another challenge is the extremely small spaces between the rotors, and between the rotors and their housing. EnginSoft invited Jan Hesse, Senior Developer of TwinMesh of CFX Berlin Gmbh, to show how designers can overcome these problems, drive real innovation, and optimize rotary positive displacement machines. Using TwinMesh Software, he demonstrated how the time-varying flow volumes and micro-gaps of volumetric machines can be automatically meshed using high-quality structured hexahedral meshes. Thanks to a special smoothing algorithm in TwinMesh, node distributions remain consistent during transient CFD analysis, preserving very high-quality mesh elements, even in the smallest spaces. This solution significantly reduces user effort and CPU time for the analysis of rotary positive displacement machines, enabling fast, reliable and efficient simulation of these machines. The keynote speaker of the event was Remo De Donno, CAE manager of Industrie Saleri Italo Spa. Based in Brescia, Saleri is a leading company in the design, development and production of hydraulic pumps and cooling systems for the automotive industry, with over 60 engineers involved in R&D. Based on the company's close collaboration with EnginSoft and Brescia University, De Donno showed how Saleri uses simulation (CFD and finite element analysis (FEA)) in every phase of product development: from concept design, to prototypes and verification of real models. The use of simulation to solve technical problems and investigate phenomena has enabled Saleri to decrease the number of experimental prototypes required, significantly reducing the time to market of their products. An expert in centrifugal pumps for the automotive sector, De Donno also showed the CFD optimization strategy being used by his team to design new pumps, starting from geometric parameterization and automatic meshing in Turbogrid, to the solution and post-processing obtained with ANSYS CFX through high performance computing (HPC). Recently, Saleri's R&D engineers have also focused their attention on the thermal behavior of electric pumps, and on fluid-structure interactions. At the end of the event, Alessandro Arcidiacono, expert of CFX and CFD for Turbomachinery at EnginSoft, produced some case studies, best practices and state-of-the-art CFD simulations using Turbomachinery. Starting with the basics (how to create a CFD model and obtain reliable results), his presentation showed the latest models available for turbomachinery (transient blade row and harmonic analysis), which significantly reduce mesh size and the time needed for accurate transient simulations. He also provided some information and examples on advanced models such as cavitation for hydraulic pumps and turbines, and aeroacoustics for fans. Particular attention was paid to the creation of parametric and consistent models involving different physics, such as fluid-structure interaction (FSI), conjugate heat transfer (CHT) and optimization. Arcidiacono demonstrated how a multiphysics platform, which can also include system-level CFD codes such as Flownex, is the best way to create a Digital Twin of a turbomachine or plant. Remo De Donno, CAE manager of Industrie Saleri Italo Spa, concludes: "The event offered the attendees the opportunity to benefit from educational presentations, the latest simulation technology, presentations on the creative application of simulation tools, and the time to brainstorm with a like-minded group during the breaks. We really want to thank EnginSoft that invited us at the workshop as the keynote speaker. EnginSoft has given us the opportunity to share our experience of fluid machines in the automotive sector, in order to demonstrate how computational simulations can become indispensable in several sectors.
The event was a success both in terms of participation and appreciation, with over 40 engineers from all areas of turbomachinery: pumps, hydraulic turbines, fans, propellers, compressors, and gas turbines."

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