Newsletter

Simulation Based Engineering & Sciences

Year 13 n°1 Spring 2016

A Great Team, Numerical Simulation and Experimental Testing @John Deere

The use of virtual prototyping tools in the design of **Nuclear Power Plants**

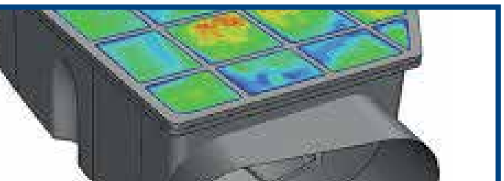
Mahindra: many companies united to enable people to rise

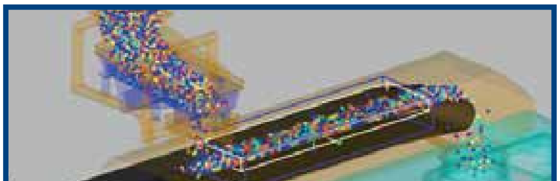
Durability and Buckling Analysis of a **Storage Tanks**

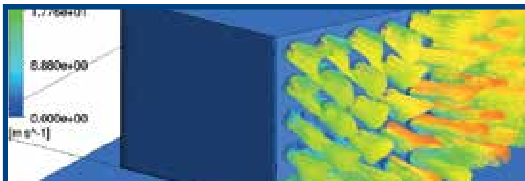
Simulation of the **transportation and handling** of granular material in a Lime Klin

Finite element analysis for **nautical purposes**

Improving the design of the **Air Purification Tower** using 3D CFD







FLASH

Intelligence is the ability to adapt to change
Stephen Hawking

At EnginSoft, the ability to adapt to the problems our customer faces has stemmed from the continuous journey of learning we have been fortunate to be part of; from a variety of research initiatives to the collaboration with many organisations, we have helped develop competitive advantage for our customers in a world of ever evolving market places. This 'intelligence' has formed the heart of our operations and remains the integral driving force.
In this issue, we explore how numerical simulation is being utilized in understanding the physical environment so that products can increasingly match the demands of where they're used. The WIN-Shoes research project, as presented on page 47, shows the way SMEs in the shoe sector have adapted by moving towards a more data rich approach to revolutionise shoe production without compromising on quality.
Whereas on page 39, we show how genetic based algorithms in modeFRONTIER can reduce a ships propeller noise so that it glides naturally in its environment and improves the on-board travelling experience.
It is without a doubt that the methods illustrated in the articles and the integration of skills in these collaborations has constructed valuable practices to adapt to future challenges.

Stefano Odorizzi, Editor in chief

Contents

OUR ACKNOWLEDGEMENT AND THANKS TO ALL THE COMPANIES, UNIVERSITIES AND RESEARCH CENTRES THAT HAVE CONTRIBUTED TO THIS ISSUE OF OUR NEWSLETTER

Newsletter EnginSoft
Year 13 n°1 - Spring 2016
To receive a free copy of the next EnginSoft Newsletters, please contact our Marketing office at: newsletter@enginsoft.it

All pictures are protected by copyright. Any reproduction of these pictures in any media and by any means is forbidden unless written authorization by EnginSoft has been obtained beforehand. ©Copyright EnginSoft Newsletter.

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

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
Cascade Technologies www.cascadetechnologies.com
Reactive Search www.reactive-search.com
SimNumerica www.simnumerica.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it

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

ADVERTISEMENT
For advertising opportunities, please contact our Marketing office at: newsletter@enginsoft.it

RESPONSIBLE DIRECTOR
Stefano Odorizzi - newsletter@enginsoft.it
ART DIRECTOR
Luisa Cunico - newsletter@enginsoft.it
PRINTING Grafiche Dal Piaz - Trento

The EnginSoft Newsletter editions contain references to the following products which are trademarks or registered trademarks of their respective owners:
ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT, FORTE', SpaceClaim and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. [ICEM CFD is a trademark used by ANSYS, Inc. under license]. (www.ANSYS.com) - **modeFRONTIER** is a trademark of ESTECO Spa (www.esteco.com) - **Flowmaster** is a registered trademark of Mentor Graphics in the USA (www.flowmaster.com) - **MAGMASOFT** is a trademark of MAGMA GmbH (www.magmasoft.de) - **FORGE, COLDFORM** and **FORGE Nxt** are trademarks of Transvalor S.A. (www.transvalor.com) - **LS-DYNA** is a trademark of LSTC (www.lstc.com) - **Cetol 6σ** is a trademark of Sigmetrix L.L.C. (www.sigmetrix.com) - **RecurDyn™ and MBD for ANSYS** is a registered trademark of FunctionBay, Inc. (www.functionbay.org) - **Maplesoft** are trademarks of Maplesoft™, a a subsidiary of Cybernet Systems Co. Ltd. in Japan (www.maplesoft.com)

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

A Great Team, Numerical Simulation and Experimental Testing

Figure 1 – Eng. Peter Pirro

Numerical simulations are increasingly used in most industries. So why do companies still spend so much money for experimental testing? I would like to give some explanations about the reasons of building expensive test equipment, which is much more expensive than software for numerical simulations, and how to utilize the different methods.

Will the customer be satisfied with the performance and the quality of the product? This is one of the major tasks during the development process. Today, numerical simulation and experimental testing are used to qualify products in different stages of the development process. Simulation models or experimental models are designed to simulate the real behavior of the product in the hand of the customer. In this context, it should be emphasized that experiments try to simulate customer usage, too. Both methods have to be validated.

In the early phase of the design, the numerical simulation has the great advantage of no need for a physical prototype. Virtual models are the prototypes of the simulation engineer. Together with the designer and other contributing members of the development team, all the existing knowledge of customer needs of the product can be included in the model. Feeding experiments are mostly needed to supply the simulation with the necessary parameters, like boundary conditions and loads. This should be done, starting on the subsystem level, depending on the knowledge of the product and the confidence level of the simulation.

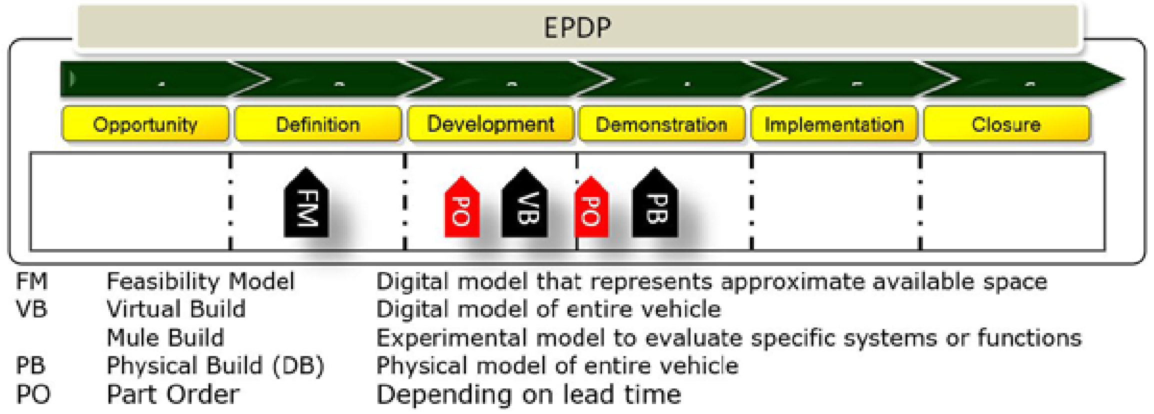


Figure 2 – The way to use simulation and experiment in the right sequence

Simulation uses basic physic principals and the used virtual model should represent the product. The results of the simulation depend the restrictions and loads. Here, a high confidence level is needed, because of the strong influence on the simulation result. Experimental tests can help to understand these effects and can increase the confidence in the output of the simulation.
If the test and the simulation engineers work as a team, both parties will benefit. Loading conditions and restrictions measured on a test bench give the input to the simulation model and the model gives back visibility of effects, that is how changes of conditions will influence the output. Experimental tests and simulations should be as close as possible to reality, to drive the outputs to be more close to real customers usage.

Especially in the early phases of the design, simulations help to sort out ineffective ideas very fast and lead to a first optimum of the product, and this without expensive physical prototypes.
The further the design evolves to the next development phase, the more physical prototypes play an important role. They naturally include a lot of special details, which are not included in the simulation model.
Some managers think that everything can be simulated, but there are a lot of arguments, that this cannot be done efficiently. Issues like the complexity of the physical interactions, the time to build and verify these complex models, and the visibility of the real influencing parameters of the product usage can be "road blocks" for "only" simulation. Complementary experimental testing is therefore the right choice. Real products have details included, which are not "designed", but produced.

How to use simulation and experiment in the right sequence and for the right reason will be explained in the next section (see Figure 2).
In my previous job, we executed a development process named Enterprise Product Development Process EPDP.

We can distinguish between two phases in early EPDP with different demands.
The first one is Virtual concept evaluation and it lasts until the Part Order or design freeze for Virtual Build. Turnaround is crucial in this phase for speed of concept generation, to maximize concept quality and to allow for DFSS (Design for Six Sigma).
Further improvements could be done by frontloading of concept evaluation to TDP/CCDP (Technology Development Process/ Critical Component Development Process) and we were working on continuous analysis efficiency improvement to gain speed.
The second phase is Virtual Verification. Competency gap closure is crucial here to reduce the need for full vehicle mules. We need high confidence level analysis and DFSS to achieve 100% Test FPY (First Pass Yield) with expensive physical prototypes.
Our approach to competency gap closure was described in the first chapter; I will now give an outlook on the continuous analysis efficiency improvement efforts.

Process improvements have already been achieved to date.
In the past, the usual builds were FB (Functional Build), DB (Durability Build) and LPB (Limited Production Build). The design, build, test, break, redesign cycle was standard. Analysis was used, but was not optimally integrated into the design process.
Today the FB is replaced by a VB (virtual build) and a Mule plus system or component tests. Analysis is an integral part of the development.
The current EPDP process allowed us to start to analyze based on the feasibility model or space claim. VB/Mule and PB have part order milestones, which we needed to take into account for the delivery of analysis results depending on part or system lead time.

Conclusion:
Numerical simulation and experimental testing can be a great team if their different strengths are used for the right tasks and in the right phase of the development.

Peter Pirro, Former Simulation and Testing Responsible, John Deere

The use of virtual prototyping tools in the design of Generation IV Nuclear Energy Systems

Ansaldo Nucleare is a company fully owned by Ansaldo Energia. The Company has the complete responsibility for the nuclear business and well established in the new international markets. In Italy, Ansaldo Nucleare is recognized as the main industrial company in the nuclear sector.

The product lines of Ansaldo Nucleare are:

Engineering and Construction of Nuclear Power Plants

In partnership with world-class established nuclear providers, Ansaldo Nucleare builds Nuclear Power Plants performing systems and main components design, core and thermo-hydraulic calculations, safety and licensing analyses, in field engineering as well as erection, supervision and commissioning. Ansaldo Nucleare has gained an extensive experience both as a supplier of NSSS (Nuclear Steam Supply System) and for architectural engineering in Italy and abroad for more than forty years.

Development of New Generation Nuclear Reactors

In the framework of the European and International co-operation, Ansaldo Nucleare develops innovative Nuclear Power Plants, offering enhanced safety features and economic improvements for the electrical production with a view to reducing the radioactive wastes generated by the plants.

Service to Nuclear Power Plants and Facilities in operation

Ansaldo Nucleare provides a large range of service activities for systems and components of Nuclear Power Plants aiming at safe operation, system optimization and performance enhancement. The Company can effectively provide these services thanks to its experience and in-depth knowledge gained in the design and implementation of different technologies for Nuclear Power Plants. Moreover, beyond design basis events and associated cliff edge effects assessments are performed to evaluate NPPs capabilities (Stress Tests) also implementing the resulting plant modifications.

Decommissioning of Nuclear Plants & Facilities

Ansaldo Nucleare has gained knowledge and experience in this field working since 1999 to pursue the decommissioning of the Italian Nuclear Power Plants and successively performing decommissioning studies and activities on Nuclear Power Plants in Europe. Activities range from the basic design to the management of on-site dismantling, including design and creation of waste management facilities.

Radioactive Waste Management

Activities performed by Ansaldo Nucleare in the field of liquid and solid radioactive waste management, cover conceptual and detailed design finalization, components and materials procurement, on-site erection and commissioning of complete systems and plants - including turnkey projects - for operational waste treatment as well as for the primary and secondary radioactive wastes generated during decommissioning process.

Figure 1 - Eng. Fabrizio Magugliani

Interview with Fabrizio Magugliani, Sr. Engineer, Aero/Thermo Analytical Design, Ansaldo Nucleare SpA

How long have you been using CAE simulation technologies and mathematical modeling in your company?

Since the very beginning of the nuclear industry, CAE simulations and mathematical models have been widespread practice. Ansaldo Nucleare has been one of the first users in Europe of 3D CAE codes. In my archive, I have a couple of documents dated early 1980s mentioning the Harwell-Flow3D CFD 3D code, originally developed at the United Kingdom Atomic Energy Authority, the 'grandfather' of the package currently known as ANSYS CFX. Since then, CAE technologies have been the standard tool for the design and validation of components and the results of the codes have been subjected to extensive V&V by the nuclear regulatory commissions.

What was the main reason for introducing these technologies?

Two main reasons have been the drivers for introducing and using CAE simulation technologies and mathematical modeling: safety and achieving optimal design. Safety is mandatory in the nuclear industry, and any design and component must be thoroughly validated against stringent safety criteria. Results of numerical simulations in specific and controlled conditions are routinely compared with properly-designed experimental results and, provided that there aren't discrepancies, the model is validated. Achieving optimal design is the other main reason: in a complex 'system of systems' like a nuclear power plant, optimization is critical to make sure that each component performs at its best and all the components together perform at their best, without compromising safety. Last but not least, cost saving is a factor for using CAE simulation technologies.

What kind of products are you using simulation for?

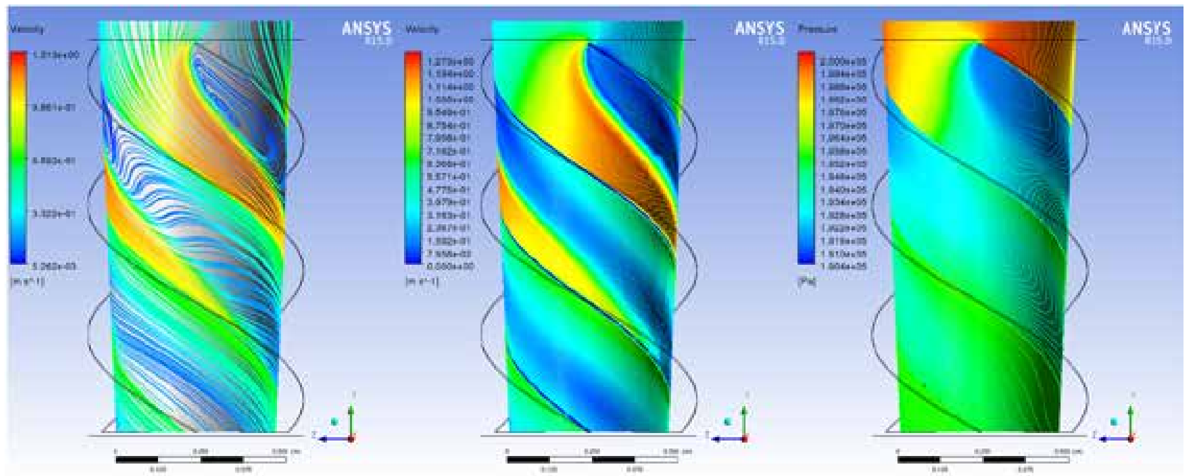
Any components of the reactor and any ancillary equipment are analyzed by itself and as an ensemble. I cannot think of a single component which does not get thoroughly analyzed in detail. The performance and operational life of the Primary Pump, Steam Generator, Decay Heat Removal System and any other components and equipment are analyzed via 3D FEM, CFD and FSI simulations taking into account not only the normal operation conditions but also any potential incidental conditions. Fluid–Structure Interaction plays a key role in the design process, because enables to evaluate the interaction between the liquid coolant and the solid components.

Why did you decide to introduce mathematical modeling in the design process?

Stringent safety requirements and the availability of versions of the ANSYS suite of packages approved for nuclear design have been the main drivers for using mathematical modelling in the design process. Beyond these drivers, 3D detailed simulations provide invaluable data for enabling the optimization of the single components and of the reactor as a 'system of systems', the assessment of the full compliancy with the safety requirements and the evaluation of the safety margin of the reactor in normal as well as in shutdown condition.

How does this affect your design process?

Ansaldo Nucleare is currently designing ALFRED (Advanced Lead Fast Reactor European Demonstrator), the liquid lead cooled nuclear



reactor, relying on the seamless integration of CAD/FEA/CFD/FSI technologies. The engineering development of the design of the reactor is dependent upon different kinds of mathematical tools: Design of Experiments, multi-scale, multi-physics modelling, Fluid–Structure Interaction and optimization algorithms. These mathematical tools supplement the data obtained with different methods in experimental campaigns. Moreover, the design team's cumulative knowledge and expertise is the most effective tool, making it possible to interpret and apply the results of 3D simulations for the development of a very effective design, leading to safe and reliable components.

Are you also thinking about applying mathematical modeling for new products and what expectations do you have?

We are exploring the merits of introducing advanced techniques for the mathematical modeling of the materials and their interaction with the liquid coolant. The modelling of the interaction between different materials and the coolant will provide invaluable data for assessing the safety of the plant in the long term, as well define the required mechanical properties for the materials to sustain in the long term the demanding environment. As of now, the modelling of the long term interaction between the material and coolant is unviable in term of computer time; my expectations are that in the future such a modeling could become a standard and sustainable design tool.

What value is EnginSoft providing to Ansaldo Nucleare?

I keep saying that EnginSoft isn't just a technology supplier but an external associate of the design team. The in-depth knowledge of the features of the FEA and CFD packages used in Ansaldo Nucleare makes EnginSoft an invaluable source of recommendations. We currently rely on EnginSoft to install on Ansaldo's computer systems the updated version of the packages and to keep our main HPC resource (a 244-core cluster) at the highest level of availability and efficiency. Moreover, EnginSoft's wide range of expertise in FEA, CFD and FSI enables a mutually effective interdisciplinary collaboration leading to a faster and error-free progression towards the optimal design of the components. Last but not least, EnginSoft makes available guidelines and real-world examples of best practices used in design environments other than the specific nuclear area that can guide the design process and avoid pitfalls and roadblocks.

In your perspective do you believe there will be a need for computation codes to handle future challenges?

There is no question in my mind about the absolute requirement to apply advanced modelling and analysis techniques in the development of current as well as future products or systems. The nuclear industry must comply with very stringent safety rules: every component must be tested and validated for the more demanding conditions and for the entire lifetime of the nuclear power plant. In the future, and according to the lesson learned with the current design practices, I envision a major emphasis on improving the overall efficiency of the nuclear power plant and on a design that allows for an increase of the operational life of the plant. To achieve these objectives, experimental campaign and 3D, FEA, CFD and FSI modelling are required, with a widespread application of Fluid-Structure Integration and material modelling techniques.

Could you estimate the return on the investment related to these R&D activities?

It is difficult to attach a monetary value for ROI when using advanced FEA/CFD design practices in the development of a complex and inter-dependent system like a nuclear power reactor. In our view, the value of FEA/CFD design practices goes well beyond any ROI evaluation, and relies on the possibility to designing a safe, reliable and efficient system of systems without having to resort to approximations or mock-up.

RecurDyn Multi-Body Dynamics boosts the Design Process of a **Planetary Stranding Machine**

About Mario Frigerio

MARIO FRIGERIO
WIRE AND ROPE MACHINERY

Since 1897, MARIO FRIGERIO is one of most reliable brand in the sector of drawing lines, low relaxation pre-stressed concrete lines, single & multi-wire treatment systems. Its machines are unique in the world: designed, manufactured and assembled in Italy, they are a benchmark in a market that rewards reliability, performance and flexibility.

Introduction

Stranding and closing planetary machines are extremely large equipment's conceived to twist, at noticeable speed, multiple heavy spools of strand, in order to manufacture large section steel ropes (Figure 1). The design of such machines has many critical aspects, mainly related the extraordinary moving masses and the potential risks deriving from a structural failure.

As sector leader, MARIO FRIGERIO has been tasked to design and manufacture a customized planetary machine, which had to meet high-demanding specifications. Accordingly to the received information, this could be the biggest planetary stranding machine ever designed in the world.

For this project, EnginSoft's collaboration has been requested to carry out a full extensive dynamical assessment of the system. In particular, EnginSoft's support had to provide fundamental inputs to the company's designers: the sizing loads and the power demand in the worst operating conditions. The task was accomplished through RecurDyn, a powerful and versatile software for Multi-Body simulation (MBS).

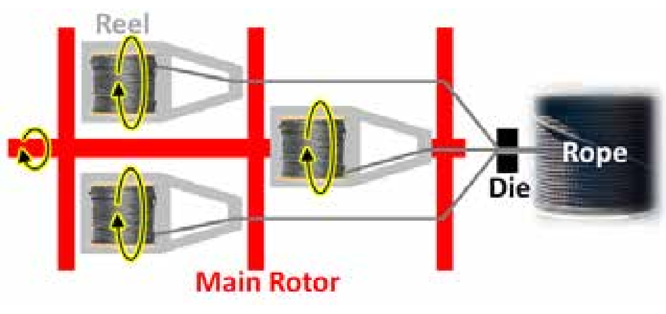


Figure 1 – Planetary stranding machine scheme

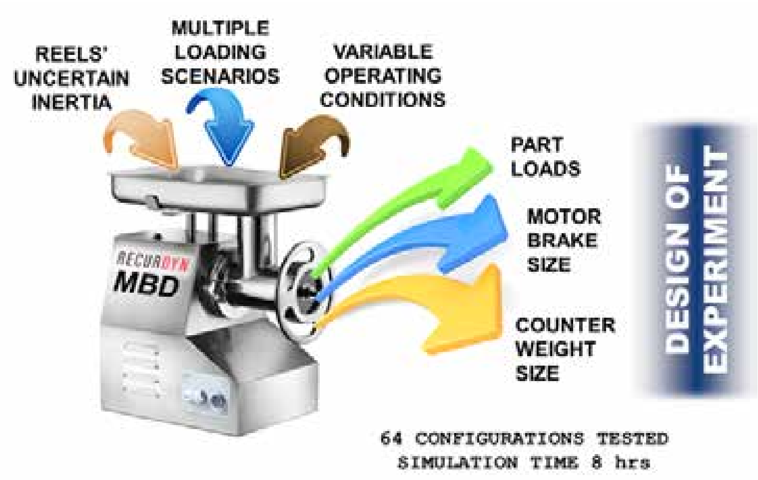


Figure 2 – DOE approach for a multi-factor dynamical problem

Approaches and methods

The final user of the machine will be allowed to manufacture different types of rope, by combining different strands in different manners. This means that the machine can be loaded by spools having different size, different position in the main rotor, different level of unbalance, and different twisting speed. The combination of so many factors lead to unpredictable loading scenarios, so that the identification of the worst operating conditions (i.e. those causing higher power demand and higher internal loads) is all, but intuitive. To provide reliable answers, EnginSoft's system dynamics experts have approached the problem through a Design of Experiments (Figure 2). First, the machine response throughout the working cycle was simulated in RecurDyn, and the analysis was repeated for all possible load sets. Then, the generated outputs, were processed all together to extract maximum and minimum enveloping curves of loads and powers.

RecurDyn Multi-Body model

The entire EnginSoft's contribution, clearly relies on the quality of the virtual model. RecurDyn software offers special features to finely reproduce the machine subsystems and to easily drive multiples simulations. Modeling operations starts with importing the CAD geometry, which includes about 100 bodies. Figure 3 shows the machine layout: it consists of multiple cages hosting multiple reels each one (the true model cannot be shown for confidential reasons). Although RecurDyn technology is very strong for flexible multibody simulation, in this project all bodies have been kept rigid. In this case, the simplification does not affect the quality of the requested outputs. All bodies are connected through different types of kinematical joints (fixed, revolute, etc.) in order to faithfully reproduce the relative degrees of freedom. Friction on rotating shafts is considered as well. All planetary gear units (Figure 4) are modeled through the coupler elements of RecurDyn. In stranding machines, the planetary gear units govern the twisting speed of all cradles. Therefore, they are the key ring in the calculation of the torque transferred the power units. By recreating the correct kinematics, RecurDyn correctly calculates the inertial reaction torques and combines them together to get the motor power.

Special attention was spent in modeling the electric motors and their control system (Figure 5). Indeed, the way the power torque is generated strongly affects the accelerations in the system. EnginSoft's expert have faithfully reproduced the torque generation in accordance with the constitutive laws of induction motors.

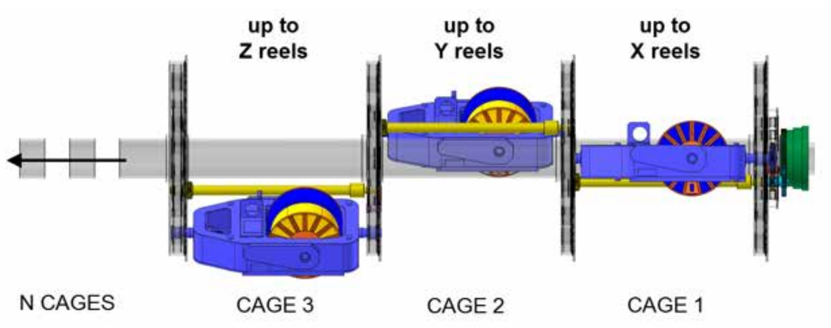


Figure 3 – Simplified view of the whole multi-body model

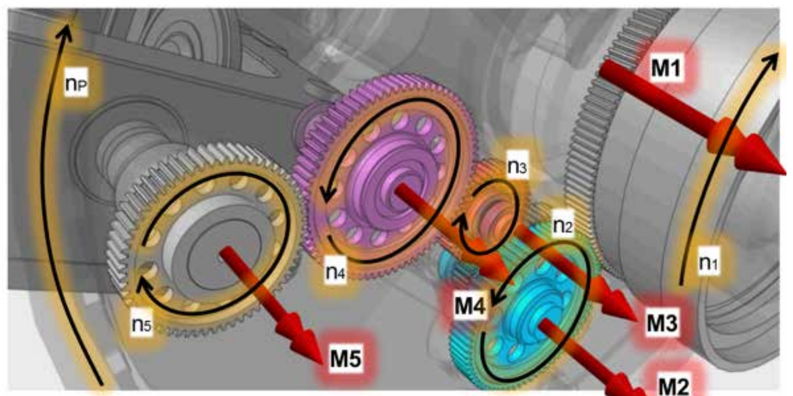


Figure 4 – Planetary gear unit

Simulation and results

For each one of the 64 load cases analyzed in this activity, the simulation covered three phases: acceleration, constant speed and emergency braking. The Figure 6 reports, as an example, how the overall torque on a driving gear looks like. The dynamics is quite severe, due to the unbalanced reels rotating at relatively high speed. All of these curves have been processed together to extract upper and lower enveloping curves. This has enabled the maximum (sizing) loads acting on each machine component to be identified, which were passed to MARIO FRIGERIO's designers for proper structural verification. At the same time, the enveloping functions of the driving torques, provided the crucial information to choose the right electric motors.

It is worth observing that choosing the motors in a project of this type is not of secondary importance. Too small of a motor would make the stranding machine unusable in some operating conditions. A motor that is too large would satisfy all requirements, but could increase the total cost of the system of an appreciable fraction. Being the power peaks strongly related to the overall dynamics, the multibody simulation is the only possible approach to avoid serious mistakes.

Conclusion

EnginSoft has supplied multi-body dynamics expertise, to support MARIO FRIGERIO's designers in the project of a large and customized planetary stranding machine. The system dynamics was accurately reproduced and simulated with RecurDyn software. MARIO FRIGERIO's feedback has proven the reliability of the simulation technology and the quality of the models used in this task. The collaboration between the companies has made possible to design the planetary stranding machine in a shorter time, while preserving the unparalleled MARIO FRIGERIO's level of quality.

Fabiano Maggio, EnginSoft

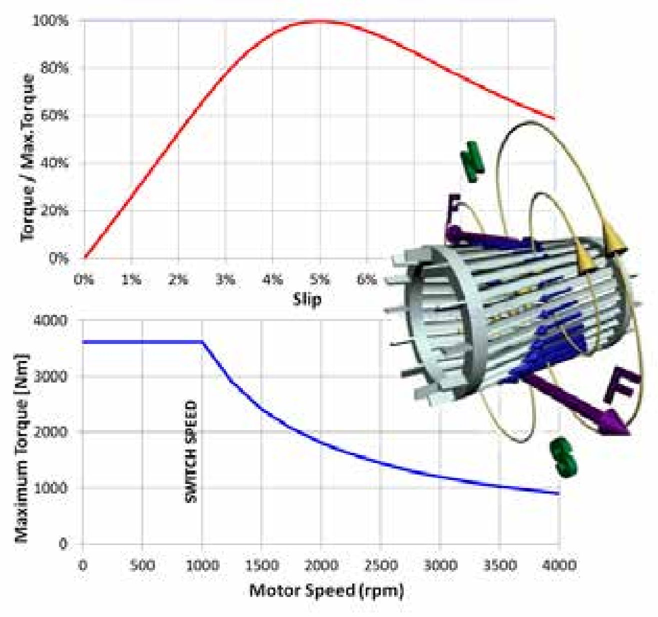


Figure 5 – Induction motor model

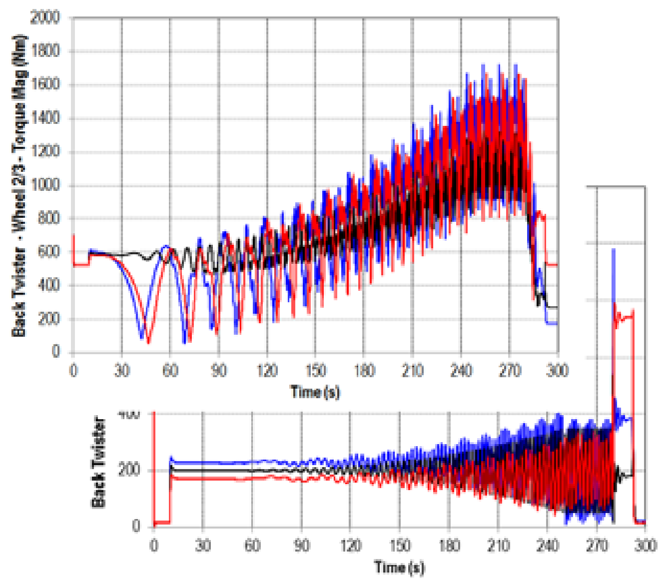


Figure 6 – Resulting torques on power shafts

The Multibody Simulation Software

The Multi-Body Simulation (MBS) is a method of numerical simulation specifically conceived to virtually reproduce both kinematics and dynamics of moving multi-body systems. Generally speaking, they are composed of bodies (rigid or flexible) interacting through kinematic constraints, relative forces, contacts, drivers and so on. MultiBody Simulation is the technology of choice for conducting motion analysis in practically any industrial sector: automotive (where it was born), packaging, robotics, appliances, biomechanics, energy.

Multi-Body Simulation is a process articulated in three main phases. First, the model is created by defining bodies and connecting them through various type of dynamic elements. Modern software strongly assists the user in completing this step, through advanced graphical interface. However, the effectiveness of a Multi-Body model strongly depends on the user capability of correctly interpreting the physics of the investigated system, before choosing the elements that virtualize it. Once the model is created, a set of Differential Algebraic Equations (DAE) is available behind the interface. The second phase of a Multi-Body Simulation coincides with the solution of those equations. Numerical algorithms, that compose the solver, are requested to integrate those equations in time domain. The solver is a crucial component of any Multi-Body software, and makes the difference in terms of performance, accuracy and stability. The third (last) phase of a Multi-Body Simulation is the post-processing. Unlike other types of simulations, Multi-Body does not return colorful images. Although it is possible to create movies of the system behavior, the most important outputs are available as time-variable signals. The weak point for most of the existing commercial Multi-Body Simulation codes is the capability to manage discontinuities (which occur when the model include many contacts) and body flexibilities (which are normally described through modal bases). RecurDyn is a modern technology, which has been developed to go beyond these limits. It manages flexible bodies using a FE-like approach, opening the door to very large deformations and true contacts over the flexible structures. This makes the Multi-Body Simulation in RecurDyn very similar to a Finite Element Transient analysis, while preserving short solution time. RecurDyn solver performs really fast even in presence of contacts and high-frequencies due to flexibilities.

RecurDyn FE-like capabilities make possible to study the overall kinematics and dynamics of flexible systems, calculating, at the same time, deformations and stresses. These strong capabilities are completed by additional functions that further extend the application field of RecurDyn: interface to control software, interface to particle dynamics, interface to computer fluid dynamics. In order to study the behavior of moving mechanisms in a multi-physics context, RecurDyn is one of the most convincing solution available in the market today.

RecurDyn is supported in Europe by EnginSoft. For more information: Fabiano Maggio, EnginSoft - f.maggio@enginsoft.it

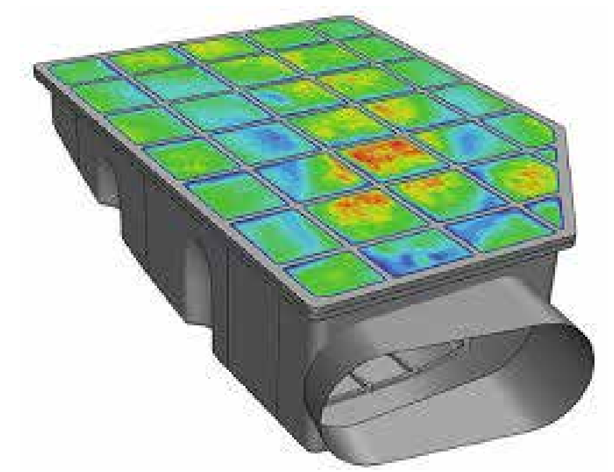
Many companies united by a common purpose: to enable people to rise

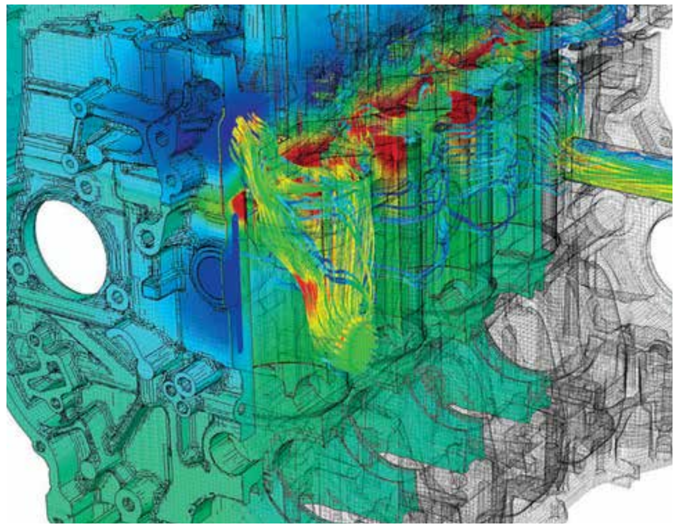
Mahindra Racing is making a name for itself in the motorcycling world, also thanks to the ambitious technological challenge of investing in an important development centre in Italy. No doubt that the international profile of MotoGP™ contributes to strengthen the image of Mahindra on a worldwide level. We have chosen ANSYS as simulation tool since it is the one better suiting our requirements and, together with EnginSoft, a qualified partner in the Simulation Based Engineering world, we aim at achieving more and more ambitious results.

Ing. Rosario Di Corte
Technical Manager - Mahindra Racing

Mahindra

Was founded in 1945 as a steel trading company and in 1947, the Group entered auto manufacturing to bring in the iconic Willy's Jeep (under license) on to Indian roads. The founders, K.C and J.C. Mahindra, believed that introducing new modes of transportation held the key to India's prosperity, so one of their first goals was to build durable, rugged vehicles that could handle the rough Indian terrain. In the 1950s and 60s, the Group diversified into businesses like Steel, Tractors, and more. Over time the Group consolidated its position in Automobiles, Tractors and Steel and entered promising sectors like IT, Hospitality, Financial Services, Components, Aerospace and Logistics, led by the goal of providing products and services that support prosperity. Today the Group is a $16.9 billion global corporation employing more than 200,000 people across the globe. The Group's flagship company, Mahindra & Mahindra, is a leader in the tractor and utility vehicles space.





CAE driven product development: how ANSYS speeds up the design process in the Moto3™ Motorcycle Racing Championship

The Mahindra Group has significantly invested in Italy in the last years, strongly sustaining the Mahindra Racing Team in the Moto3™ class of MotoGP™ World Motorcycle Racing Championship. The technical team is composed by determined people with high prospects for the future. The Mahindra Racing's engineers are focused on R&D activities and production: in this scenario the use of simulation becomes strategic because it allows to increment, fasten and make the communication between development and production more efficient. The choice to employ the ANSYS technology realizes this vision, the Workbench environment stands out for its robustness and accessibility. The current challenges are the weight reduction of the primary structures (i.e. chassis, fork) and the development time reduction for new components. Moreover, being necessary to adapt the motorbike to the racetrack every time, the main design parameters are re-set in order to pursue from time to time the best configuration. Today the numerical simulation offers an important added value for dimensioning and testing of the motorbike's chassis, fork and power train (in particular for the intake valves); after the acquisition of experimental data on the stiffness of some components, ANSYS has been essential for the numerical-experimental correlation.

ANSYS in Motorsports

Because of their competitive nature, motorsports are the testing grounds for pioneering automotive technologies. Hybrid and electric drives for race cars pose engineering challenges along with the opportunity to lead the pack. Furthermore, motorsports regulations have tightened the use of wind tunnel testing and virtual simulation. In Asia, where two-wheeler use is on the rise, engineering issues include improving fuel efficiency and reducing pollutant emissions. There is a growing market for electric drive two-wheelers. Of course, innovation, quality, reliability and safety are paramount.
Engineering simulation is the answer to fast exploration, development and optimization in this sector. In motorsports, race teams must upgrade virtual product development methods to the most advanced, accurate, efficient and scalable tools and techniques. Motorcycle manufacturers can leverage simulation technology to innovate engines, transmission and electric drives while operating within tight cost constraints. Electric drives involve interacting thermal, fluid, structural and electromagnetic effects that call for comprehensive multiphysics simulations, which is an ANSYS hallmark. For motorsports pioneers, high-performance computing options and efficient, scalable solvers mean you can get the most out of the limited computer flops that motorsports regulations allow. Motorcycle manufacturers can leverage our design exploration and optimization tools — such as persistent parameterization, adjoint solver and morpher —to implement early quality and reliability prediction methods.
For more information: Fabio Rossetti, EnginSoft - f.rossetti@enginsoft.it

Lime Kiln DEM Analysis with ROCKY

To remain a world leader in the lime industry, Cimprogetti uses the latest and most advanced technologies: ROCKY Dem has proved to be the best solution for simulating the transportation and handling of granular material with different shaped particles. With technical support from EnginSoft we have developed a project that can analyse and optimize limestone dynamics inside a lime production kiln. The results have been excellent, and future experimentation in improving the efficiency of our plants will definitely proceed.

Eng. Valerio Colombari
Head of New Product Development Unit
Cimprogetti

Cimprogetti was founded in 1967 by Pierluigi Rizzi, former President and current Sole Director, in Bergamo (Italy), the 2nd most important industrial province in Europe, located in Lombardy, a region rich in tradition and expertise in the production of lime, cement, ground carbonate, etc.

The entrepreneurial skill of the company's founder, along with the experience and determination of the staff, has allowed Cimprogetti to rapidly expand its operations into sixty countries worldwide, spanning the five continents.

The company's product range spans from vertical kilns for the calcination of limestone and dolomite to complete quicklime hydration units. Over several decades, Cimprogetti has consolidated its distinctive business approach: to provide comprehensive lime plants from A to Z, from know-how and engineering design to an EPC basis delivery and to supply innovative and high-performance plants based on well-proven design, adopting tailor-made solutions whenever needed, to meet the customers' specific needs.

Process Engineering is vital for any industrial plant development. Its broad scope ranges from site visits, surveys and discussing issues with operators, to long-term project development. Our Process engineers design and optimize chemical processes, advise on process safety issues, analyze and interpret laboratory and plant data and, generally speaking, provide technical support to customers with the aim to ensure the technical integrity of equipment and plants. The Lab forms part of our design department and is equipped with state-of-the-art 3D CAD and CAE workstations to support all the engineering phases of a project. The use of simulation tools allow to predict the behavior of design solutions. It is possible prove innovative concepts and optimize designs, starting from the early stage of the design and engineering process until to the product validations and to a better understanding of the implications of design decision on the plant performances.

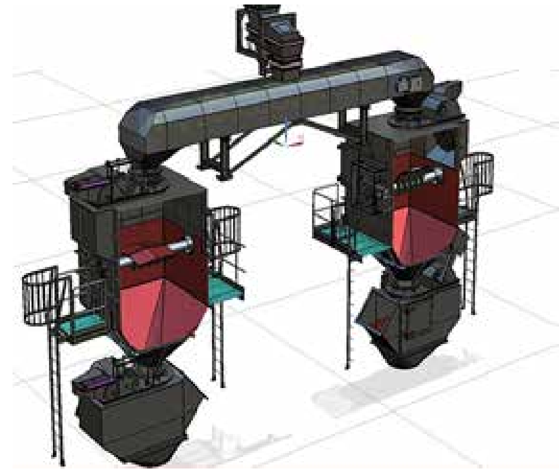


Figure 1 - Lime Klim model

The use of ROCKY DEM in design

Cimprogetti is a worldwide leader in designing equipment and plants for the lime industries. As far as lime kilns are concerned, they can be located in different plant configurations, such as chemical plants, construction sites, steel plants or lime producers. Typically, the handling and the storage of the lime that feeds the kiln, it will vary continuously according to the customer's specific plant requirements, whereas the load part on top of the kiln remains unchanged. This is organized in a way that manages and optimizes any lime distribution situation coming from the kiln upstream. The aim of the activity carried out by Cimprogetti in collaboration with EnginSoft, was to perform a dynamic analysis of the kiln's lime feeding process, starting from the typical plant configuration with skip fed by a weighing hopper, until the charging of limestone into SRT hopper of the new kiln loading system designed with the last Vanguard® kiln technology .

The tool used for the analysis is ROCKY, a DEM (Discrete Element Modeling) code. The software, starting from the CAD provided by Cimprogetti, has allowed us to virtually reproduce the whole lime transportation phase, handling the skip, the conveyor belt and the internal flap of the SRT hopper. An important part of the simulation has been the calibration of the particle's characteristics, both physical (friction and impact) and geometrical (real and not spherical shape) ones. The simulation has showed how the limestone reach the top of the kiln and has therefore allowed us to improve the design of kiln loading system with the result of a uniform stone distribution.

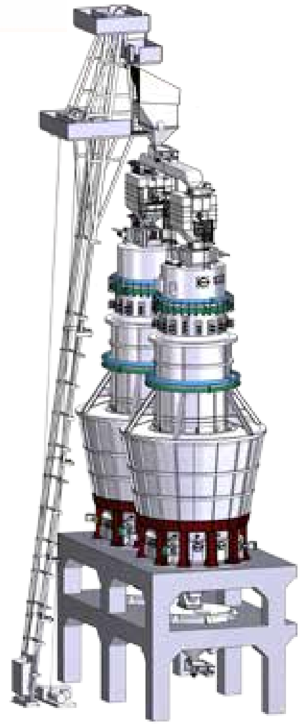


Figure 2 - Lime klim

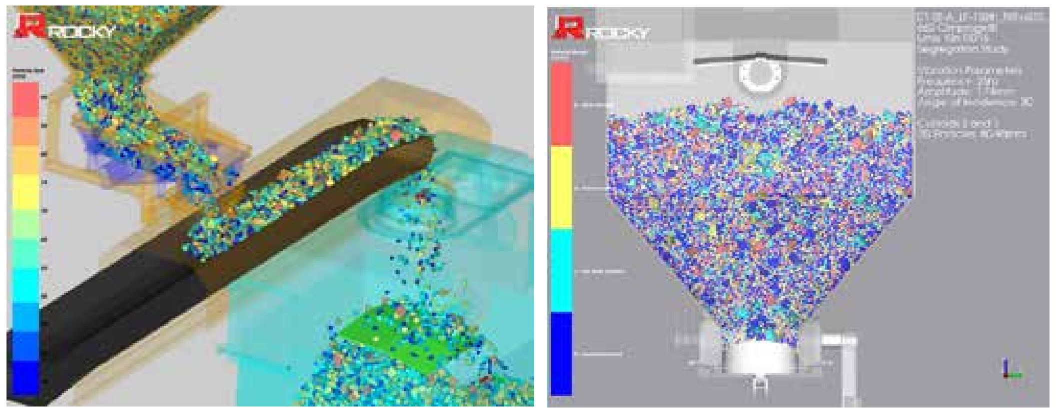


Figure 3 - Rocky DEM simulation

ROCKY is a powerful, 3D Discrete Element Modeling (DEM) program that quickly and accurately simulates the granular flow behavior of different shaped and sized particles within a conveyor chute, mill, or other materials handling equipment.

Several capabilities set ROCKY apart from other DEM codes, including non-round particle shapes, the ability to simulate particle breakage without loss of mass or volume, the visualization of boundary surface reduction due to wear, and more.

ROCKY is supported in Europe by EnginSoft. For more information: Massimo Tomasi, EnginSoft - m.tomasi@engisnoft.it

Case Histories

Durability and Buckling Analysis of Storage Tanks

Teknokon Group is a multi-disciplinary organization headquartered in Istanbul, Turkey. Its holding, Teknokon Machinery, founded in 1993, is an engineering and manufacturing company with wide international experience. They provide mechanical design and production of various process equipments used in sectors such as chemical, petrochemical, energy, mining, food, beverage and cosmetics, in accordance to the certification of quality, environmental and occupational health and safety management systems regulated by the respective standards (ISO 9001, ISO 14001 and OHSAS 18001).

Project Objectives

Teknokon Machinery has developed its know-how in design and manufacturing of various types of storage tanks during the years and they have become among the leader in this field. By taking advantage of EnginSoft experience in Simulation Based Engineering, by means of the FEM analysis, the Durability, Structural Integrity and Buckling behavior of a new group of toxic liquid and fuel storage tanks have been investigated..

FEM analysis to investigate and improve the durability of products

In this project, thirteen different kinds of storage tanks have been modeled using ANSYS Mechanical. Three dimensional geometry and the finite element models have been generated from 2D technical drawings in order to investigate the mechanical behavior of the tanks when filled with liquid and to predict the buckling behavior when they are empty. Two of the storage thanks had a mixer located at the top of the roof with an installation platform. The mixer has been considered as a lumped mass rigidly connected to the platform.

Quadratic shell elements with full integration option have been used to mesh the model. Converged mesh has been used in critical areas where the stress gradients higher to be able to capture the right mechanical response . Loads acting on the system have been provided by Teknokon. Forces and moments have been applied to the flanges and hydrostatic pressure has been applied to the inner surfaces of the tanks. The effect of gravity has been considered: snow loads, presence of the weight of people and the wind forces have been taken into account in the finite element analysis. Fixed supports have been introduced through the baseline of the models.

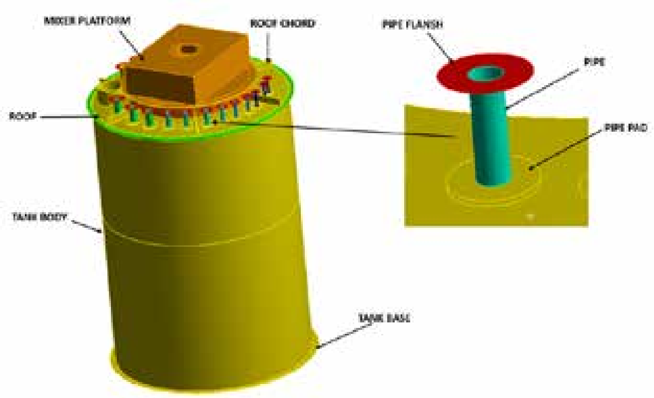


Figure 1 - 3D geometry and parts of HX62-CM002 tank

Figure 2 - Mesh

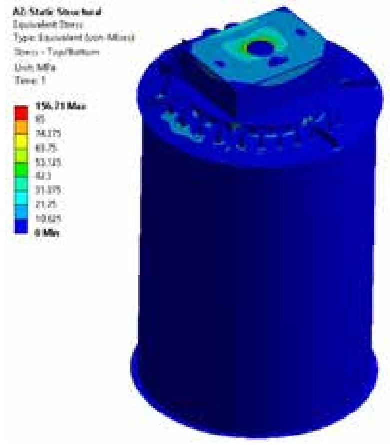


Figure 3 - Results

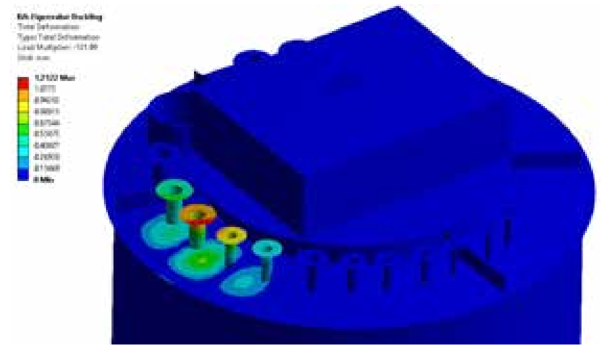


Figure 4 - Detailed Results

Conclusion

FEM analysis has allowed us to better understand the mechanical and buckling behavior of the structures. In this way, we could define the proper geometrical modifications necessary to guarantee the structural integrity of the storage tanks.

For more information:
Tahir Soyugüzel, EnginSoft Turkey– t.soyuguzel@enginsoft.com

EnginSoft Turkey: the business is growing

EnginSoft Turkey started the activities in February 2015. The offices are located in Teknopark Istanbul and Ankara. In less than a year, the subsidiary has been growing its business with important projects and partnerships. One of these projects, reported in this newsletter, is about the activity with Teknokon, an key Turkish player in the field of oil&gas.

The company provides mechanical design, residence engineering, advanced engineering analyses, software sales and technical support services and carries out co-funded R&D projects. EnginSoft Turkey is currently providing solutions in aerospace & defence (A&D), automotive and appliances sectors. Their aim to be a solution partner of the main players in these sectors.

EnginSoft Turkey has introduced and is helping many companies adopt modeFRONTIER optimization and Cetol 6σ tolerance technologies. EnginSoft Turkey is the sole distributor of modeFRONTIER (optimization platform) and Cetol 6σ (tolerance analysis) in Turkey. Furthermore, EnginSoft Turkey distributes Flowmaster, Recordyn and LS-DYNA and provides for these softwares technical support. Our expert staff has worked intensively in our first year and has recently organized modeFRONTIER and CETOL 6σ seminars to introduce softwares with a high number of participants from leading companies in a wide range of sectors in Istanbul and Ankara, also, EnginSoft Turkey regularly arranges webinars, software trainings and special trainings about the usage.

We are extremely please with the reception from Turkey and look forward to future collaborations especially in R&D and upcoming A&D projects.

For more information: Sadi Kopuz, CEO EnginSoft Turkey
s.kopuz@enginsoft.com

MICAD, engineering aimed at the sea's jewels

During my professional career started at MICAD, a yachts engineering and design studio, I challenged myself working on different difficulties projects of several lengths boats. Our goal is definitely to be the beachhead between traditions in naval constructions and innovations developed by the latest technologies in 3D modelling and numerical analysis software for CFD as well as FEM

Daniele Bruno
Numerical Analyst, MICAD

MICAD has begun his activity since 2004, the expertise in naval, mechanical and design fields of the three founders were merged to create a center of competence suitable to collect energies coming from different grounds. We are rapidly shooting up and now the team is made up of 11 people.

Our grounds are the naval and nautical fields focusing on the engineer consultancy for shipyards all over the world.

We have been working since 2010 alongside the novelty in the worldwide panorama of luxury yachting, the brand "Monte Carlo Yachts".

They are pioneers in nautical composite materials constructions introducing for the first time the modular assembly system, for this reason they gained in 2015 the "innovation in a production process award" as a part of the prestigious new Boat Builder Awards for business achievement organized jointly by IBI magazine and METS. MICAD is partner of Monte Carlo Yachts assisting them in their growing up.

We have also established collaborations with the giant boat builder Bénéteau, developing the naval architecture of Monte Carlo 4, Monte Carlo 6, of the last two models launched in the "Gran Turismo" gamma, the GT40 and GT46, as well as the new Swift Trawler 30.

Figure 1 – SpaceClaim geometry relative to a part of the structures of a yacht 30-meter length

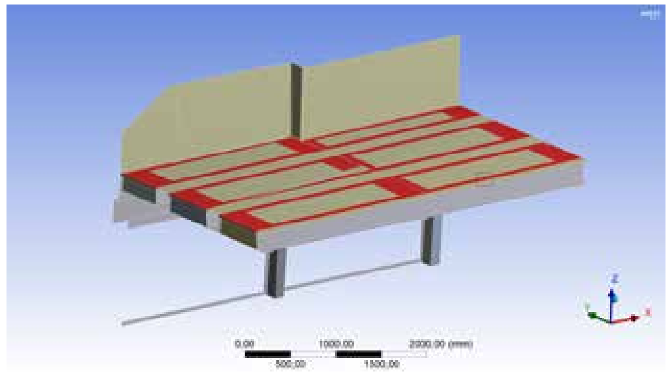


Figure 2 - Closed volume, representing the glue volume of 15 mm thickness

Traditional naval architecture meets the CFD analysis with the aim to improve consolidated products reaching values in terms of performance never seen before.

Tradition meets innovation

Modular construction system, as well as other important innovations, is game-changer for the industry giving the possibility, for example, to cut construction time; anyway this means investments.

Naval industry has not the same capacity to invest as the automotive or aerospace ones and this means to be often a step back on the side of technical innovations.

Our goal is definitely to be the beachhead between traditions present for centuries in naval area and innovations coming from other sectors with higher quality series productions.

To transform an idea into reality is a hard task, there are a lot of small troubles that come out during the engineering process; working in close contact with real problems, we have developed a unique proficiency in helping industrial manufacturers replacing physical prototypes by virtual ones, assembling and testing products in different environments.

Virtual prototyping enables our clients to evaluate the performance of their product and the consequences of its manufacturing history, under normal or accidental conditions.

By benefiting of this information early in the process, enterprises know whether a product can be built and whether it will meet its performance and certification objectives, before any physical prototype is built.

To enable customer innovation, MICAD's solutions integrate the latest technologies in 3D modelling and numerical analysis software for CFD as well as FEM.

With this vision we are now partner of EnginSoft.

Computer-aided engineering

Nowadays, CAE approach allows us to work close to the production having an augmented reality on numerous aspects of the product process. Innovative 3D modelling software, like SpaceClaim allows us to find instantly for example interferences or if there is enough space for the systems and equipment.

FEM and CFD work together to simulate which could be the impact of a change about any part of the project, giving advices on the stiffness of the structures for example loaded by hydrodynamic pressure.

The whole is computed fast and with a great precision, but overall software like ANSYS Mechanical, allows to have constantly a graphical view on the product in terms of 3D viewing on which users can add virtual information calculated in FEM or CFD environment.

Nautical structures scantling

For naval applications, scantling is performed through the international ISO-12215 rule (boats under 24 meters) or rules coming from the national classification societies that essentially give to the designers standard loads to assign to each structure or panel, several minimum scantling values (minimum thickness, section modulus, etc.) and criteria by which define materials.

For the case study, core of this publication, we have faced with R.I.NA, the Italian classification society for ships. Even if the modular approach of the construction process by Monte Carlo Yachts was already certified for boats under 24 meters, for the first vessel classified as "ship" of the brand (MCY 105), we had to figure out the requests of the register, checking through numerical analysis, the strength of the gluing between the fly and the deckhouse modules. The MCY 105 received "the Most Innovative Yacht Trophy - Cannes Boat Show 2015"

Case study

The biggest challenge I had was the study of the gluing among structures cause of the difficulties to model the non-linearity behavior of the adhesive.

In the image 1 is showed the case study relative to the gluing of a fly deck with the underlying structures of deck house for a yacht of 30-meter length.

The goal was to understand the behavior of the adhesive among the parts in order to verify calculations from the rules adding the contribution of gluing.

The ANSYS package provides the 3D environment SpaceClaim by which we can define the geometry, splitting or joining surface patches for example. The software is also capable to find out several faults in the mathematical definition of the shape, non-

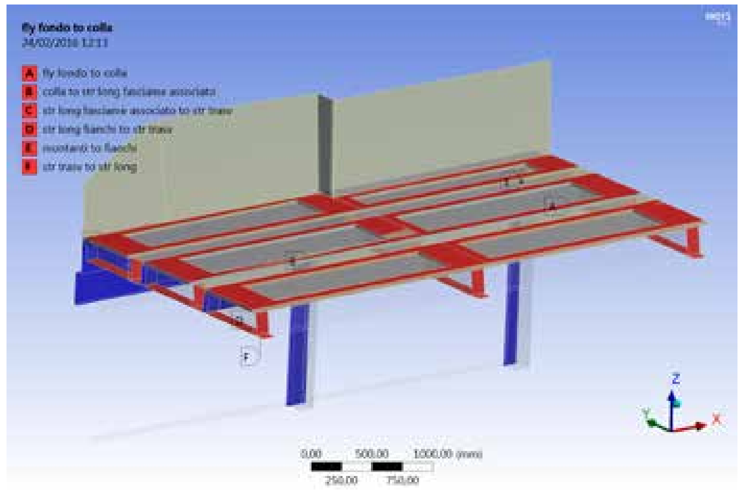


Figure 3 - Mechanical allows to insert connections among parts defining how parts interact with each other

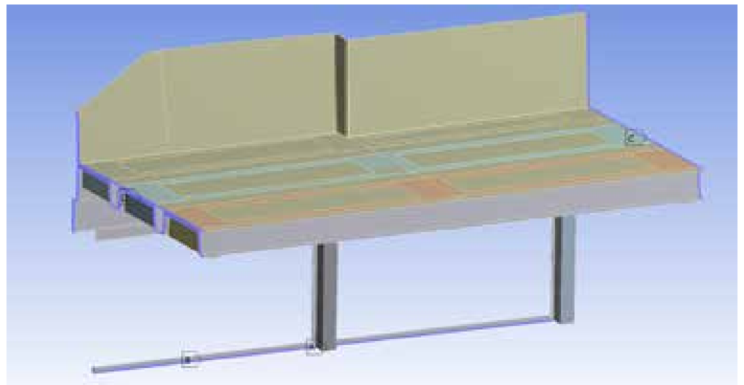


Figure 4 - Bases of struts are fixed by displacement bounds; the aft, the fore and the bottom contour are simulated by fixed supports on their edges

manifold edges as well as inconsistent surface normal direction.
As common workflow we usually import surfaces from CATIA or Rhinoceros and we prepare the surfaces for the next FEM simulation inside SpaceClaim environment.
The first phase was modelling the geometry simplifying the shape to build a handy geometric model, in this phase is also included splitting the geometry parts in order to recognize the surfaces on which is necessary to create a lamination and the surfaces that identify closed volumes.
The latter is useful to represent solid materials like PVC used to simulate the core for sandwich structures. Image 2 shows some 3D closed volumes representing the glue volume, with 15 mm thickness, at which we refer the material characteristics of the Crestomer adhesive.
Crestomer is a material provided by the well-known Scott Bader company, its mechanical characteristics come from the software EsaComp, another software we have at our disposal.
In Mechanical it can be possible to assign to a volume the characteristics of a specific material and on his faces the relative laminations.
About laminations, the software permits to assign the layer stratification starting from a reference surface, for this case study it was considered the external surface and the order of the layers is assigned starting from that.
This step is quite important and it must be done with great attention in particular to the directions of the layers that need to be oriented to form coupled layers. It is important, for example in a quadri-axials weaving, to set four layers oriented in + or – 45°.
Next important steps of this FEM simulation are the connections definition and the bounds assignment.

Simulation settings

As shown in image 3, the item is a quite complex one, it is composed by numerous parts and several laminations. Mechanical allows to insert connections among parts with the aim to create the correct physical model defining how parts interact with each other; several types of connection are available: bonded, no separation, frictionless, rough and frictional.
The first two types are linear, they need only one iteration to be solved, last three types are non-linear and require multiple iterations.
To be defined, one side of a contact pair is referred as the "contact" and its mate as the "target", Mechanical uses a color coding system to differentiate the contact and target surfaces.
For this study case we mate the gluing volume: its top face with the bottom one of the fly floor, and its bottom with the top face of the associated plating of the deckhouse' structures.
The other contact regions are displayed in image 3.
About supports, it is important to decide how many degrees of freedom is necessary to leave to the structure, with fixed supports is possible to constraint all degrees of freedom (dx, dy, dz, rotx, roty, rotz) on vertex, edge, or surface, with displacements is possible to allow for imposed translational displacements in x, y and z.
In ANSYS environment there are a lot of linear and non-linear supports that can help operator to identify the behavior of the physical system.

In this case study on the structure, as shown in image 4, four zones were fixed. Firstly, the base of struts through their bottom faces are fixed by displacement bounds; then, the aft, the fore and the bottom contour are simulated by fixed supports on their edges.
About loads, it is possible to define several kind of loads in different parts of the structure like edges, nodes or surfaces.
Two loads conditions respectively with a vertical pressure were applied on the floor of the fly deck and the second with an acceleration applied on the entire structure. The vertical pressure case was taken into account as base to simulate the behavior of the structure under the static loads of its mass plus all the effects and people that will be taken on board.
The case in which we are added the accelerations was useful to investigate the reactions of the structure weighed down by the loads of vertical and lateral wave motions simulated through vertical and lateral accelerations.
The symmetry was taken into account to reach the complete model holding-down the number of cells for the simulation. Symmetry condition was added to the central contour edge in order to make the structure symmetrical.

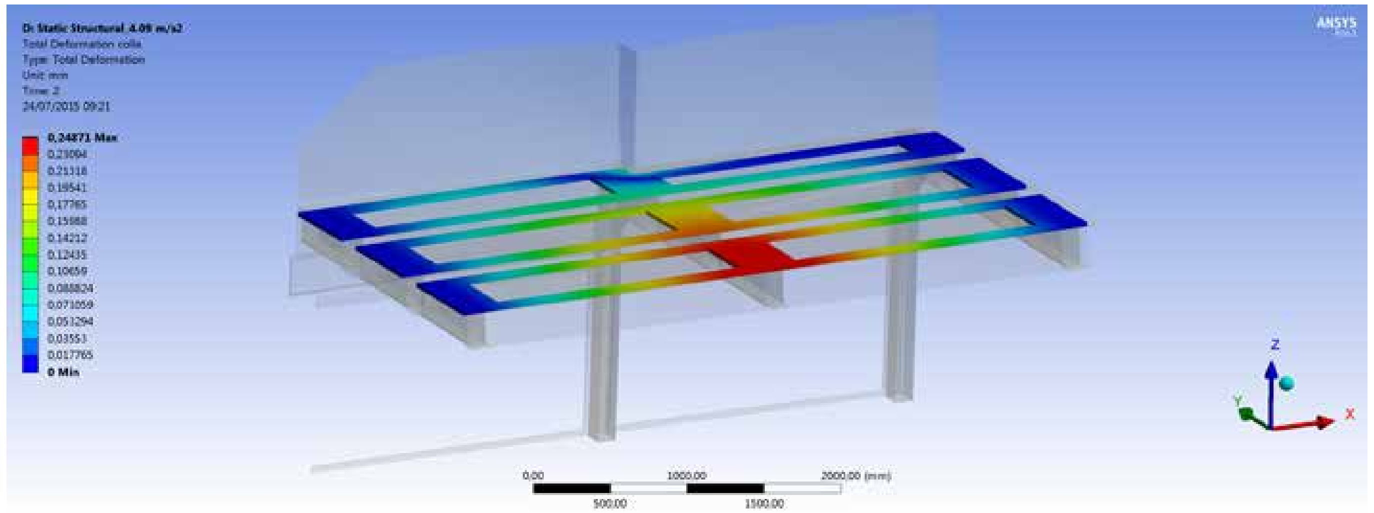


Figure 5 - Only 0.2 mm deformation at the centre zone of the 15 mm glue inserted to join the fly to the deckhouse structures

Results

After the definition of the simulation, that means after the pre-processing phase, the solver could make calculations on the discretized domain in order to report the values of the interested sizes for each node. In the post-processing phase, colored maps representing the size of interest are made in order to understand what happen in the structure like deformations or inner stress.
In image 5 is shown the deformation of the 15 mm glue inserted to join the fly to the deckhouse structures. Is easy to see how could be thin the deformation with a maximum of 0.2 mm in the center zone of the fly floor. The image is only an example of how many possibilities ANSYS has in order to perform a good post-processing phase; in this case only the deformation caused by a vertical pressure was displayed.

Conclusions

Many other tips or results could be presented in these pages but there is one important thing to remember: software are tools. Behind them there is a team of engineers and technicians.
MICAD has developed a close partnership with EnginSoft to expand our collaboration activities in other grounds and in order to share our expertise in naval and nautical fields with them.
We believe this is the only way to help our clients to get over new challenges.

Daniele Bruno, MICAD

Image at the top and at the bottom of the article:
Courtesy of Monte Carlo Yachts

CAE technologies in the Marine Industry

The Yacht Design world changes faster than even before due to the new and high expectations of the customers. To stay competitive, it is necessary to keep up with the latest trends trying to mix the distinctive features required by customers, like functionality and reliability, and needs of innovative and stylish design. In order to develop a successful business strategy, the Concurrent Engineering can play a key role.
This new method is based on a cross functional approach which changes the traditional workflow between production process and design development. By the way, the MICAD experience confirms that ANSYS and ESAComp are essential to reach the success.
The multidisciplinary groups act together overcoming the areas of major concern such as cost-saving, high performances and speed.
The new generation CAE technologies have a large impact on this scenario providing perfect integration between engineering application and innovational stylish.

For more information: Fabio Rossetti, EnginSoft
f.rossetti@enginsoft.it

Improving the design of the Air Purification Tower using 3D CFD

The Nederman Group is a world leading supplier and developer of products and solutions within the environmental technology sector. Their latest step is the use of 3D CFD which improved the performance of their Air Purification Tower by 15%!

Smoke, fumes and particles all have a negative impact on your working environment thereby affecting production quality and profit. The most effective method of controlling welding fumes is extraction at source. This is however not always possible and sometimes insufficient. In such cases, Nederman offers the Air Purification Tower

Nederman's Air Purification Tower is the perfect choice if source extraction is not a satisfying option. Such conditions can for example be found in large workshops with changing activities and where ducting cannot be installed. The Air Purification Tower can also be a complement to existing Nederman source extraction.

Figure 1 - Preferred welding smoke extraction: Nederman's extraction arm to extract the harmful welding fumes at the source

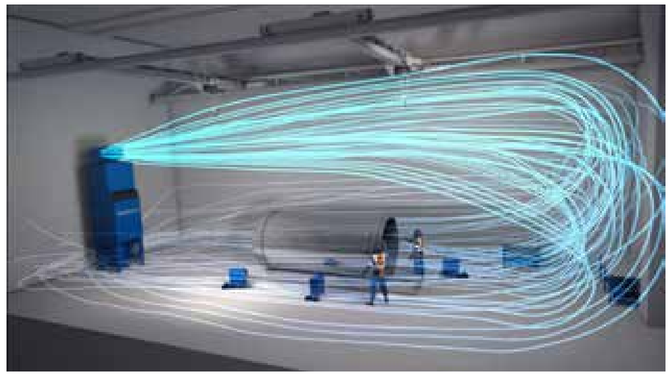


Figure 2 - Nederman's Air Purification Tower extracting harmful emissions, when it's not possible to extract at the source

By injecting air at a high velocity and at the correct altitude the Air Purification Tower sets the air in the room in to motion, see figure 2. The height and speed of air effectively removes the grey cloud of welding smoke that can form in workshops or production areas. The polluted air is extracted into the filter at the bottom of the Air Purification Tower. This means that the system continuously filters the polluted air and the filtered air is re-circulated back into the premises.

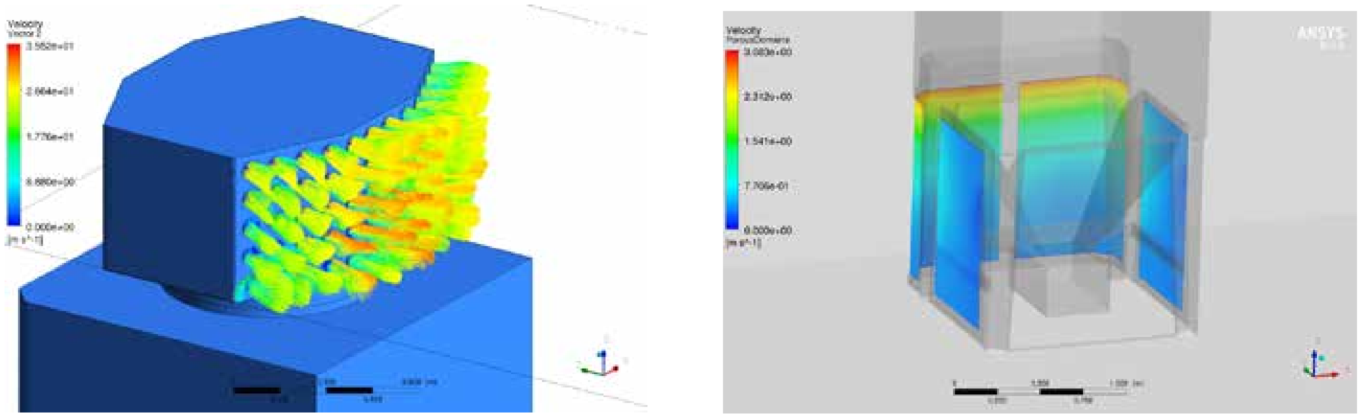


Figure 3 - To the left the injection area is shown, with vectors showing the direction and magnitude of the flow exiting the Purification tower. To the right the extraction areas can be seen, where a velocity distribution has been plotted on the walls of the perforated plates

When designing the new Air Purification Tower the target was not only to improve product performance, but also to improve the understanding of how the placement of the unit in the workshop can be optimized and how multiple units can create synergetic effects improving capturing efficiency.

Aim of the project

The aim of the CFD study was to build a model that is able to support optimization of the design and placement of the Air Purification Tower in real working environments, with the final goal of maximizing the particle separation efficiency. The project was divided into two different phases, the design phase and the room placement phase.

Phase 1: Design

When looking at the design of the tower, two main areas of interest could be identified having the biggest impact on the overall performance of the particle separation efficiency. These areas are the injection area and the extraction area, see figure 3.

Simulations were performed comparing the particle separation efficiency depending on different designs of the injection area and extraction area. By optimizing the design the particle separation efficiency could be increased by 15%.

Phase 2: Room placement

To gain a better understanding of the effects of the placement of the purification tower, several simulations were performed, modifying the dimensions of the room, the placement of the tower as well as the number of towers in the room. In figure 4 one of the multi unit simulations for a large production area can be seen.

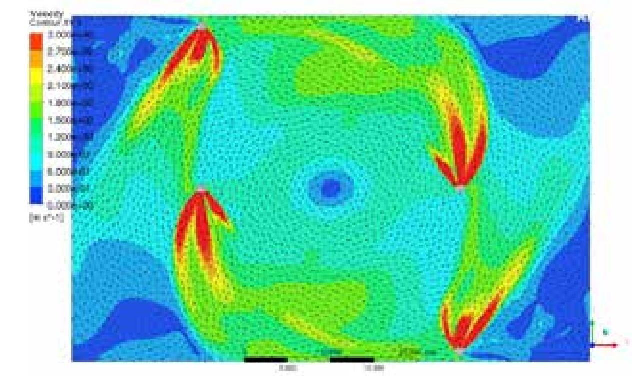


Figure 4- Interaction between 4 units in a large size production area

"By using CFD simulations we have been able to optimize the design and understand the important factors affecting particle separation from the work environment. In addition we were also able to create animations of the air stream behavior that was of great value to our sales staff"

Christian Norman,
Director Product Development Nederman

Figure 5- Nederman's new Air Purification Tower

For more information:
Christoffer Järpner, EnginSoft
c.jarpner@enginsoft.se

Mixing simulation in manufacturing process of cosmetics in SHISEIDO

Shiseido Company, Limited is the largest cosmetic firm in Japan. Since its foundation as Japan's first Western-style pharmacy in Tokyo in 1872, Shiseido Group has expanded its business not only in cosmetics but also in various domains and has led the Japanese cosmetics industry and culture for over 140 years. With the creation of beauty and wellness as a mission, Shiseido Group is a multi-brand company which operates its businesses all over the world including Europe, United States and Asia.
The technology & engineering center of Shiseido started using the MPS based simulation software Particleworks in 2012. In this article, we introduce a practical application by using Particleworks for the evaluation of material mixing.

In the early stage of development, new cosmetic products are experimentally produced in a laboratory using simple equipment. However, it may be difficult to manufacture qualitatively the same product in a production scale as the dimensions and the structure of the mixing system are very different from those of the laboratory. So the optimal design of the mixing system based on numerical simulation is required. An example of the mixing system is shown in Fig.1. The state of mixing in the mixing system was simulated and compared with experiments. The simulation was performed with 129,269 particles as 3.5 liter of fluid and 185,185 particles as 5.0 liter of fluid.

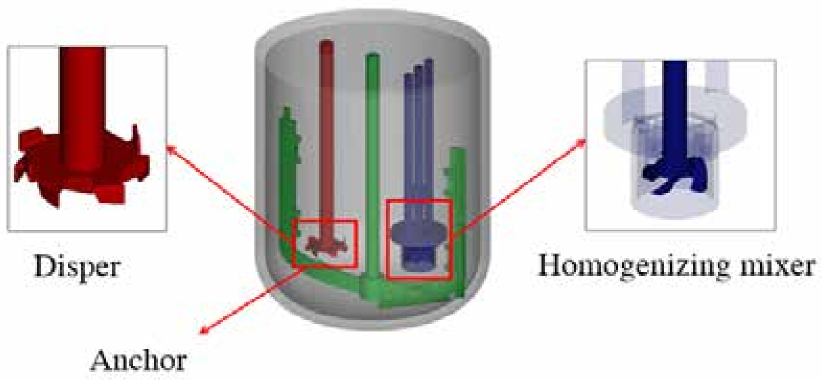


Fig.1 - Example of the mixing system for emulsification and dispersion

The mixing of fluid A using the anchor with a rotation speed of 90 rpm is shown in Fig.2. The comparison between the simulation and the experiment for the depth of the free surface in several conditions is shown in Fig.3. The simulation results are very close to the results achieved from the experiment.
After the verification, a Particleworks simulation to predict real manufacturing process of ten ton material was performed. Two cases with conventional blade and newly designed blade were compared. Flow velocity distribution of both cases is shown in Fig.4. A small convection can be seen on the left hand side of Fig.4 (a) and an advection at the bottom right corner is also shown for the mixing using the conventional blade. In contrast, a large convection in the container can be seen for the mixing using the newly designed blade in Fig.4 (b).

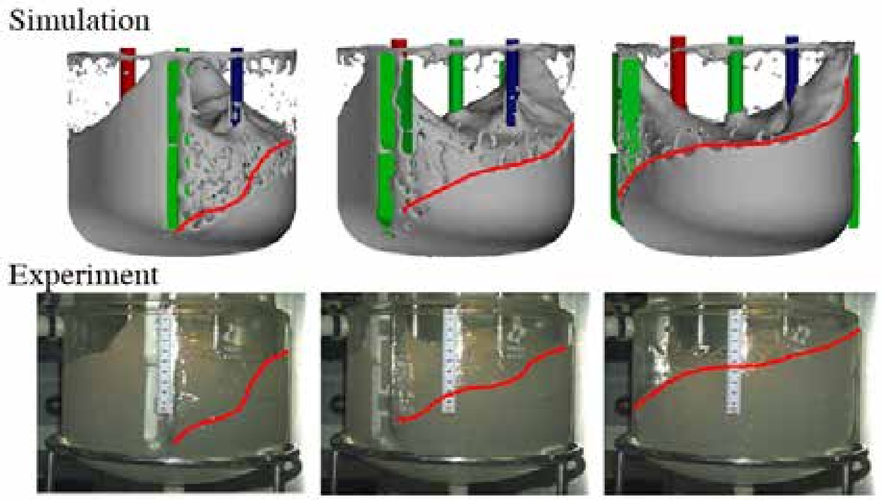


Fig.2 - Comparison of mixing states of 3.5 liters of fluid A with an anchor rotation of 90 rpm. The red line shows the depth of free surface at the rear side of the anchor

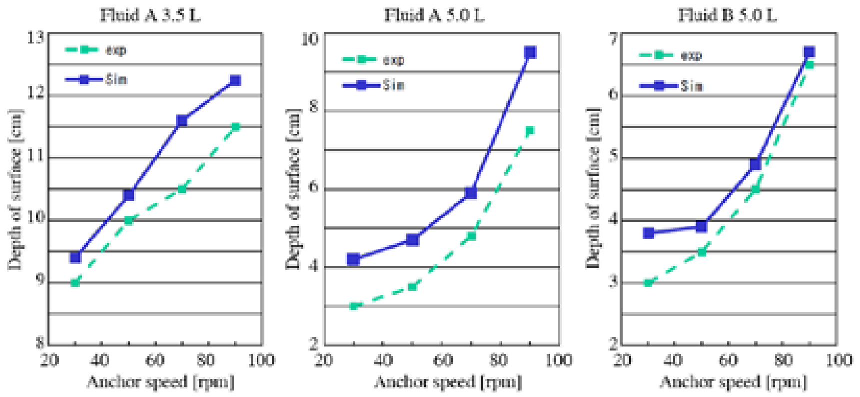
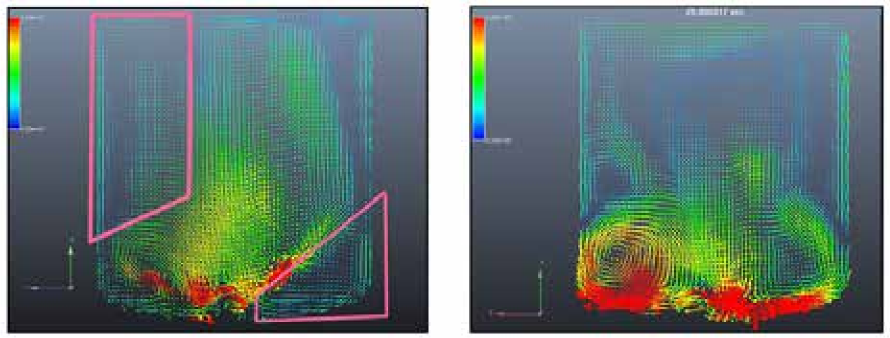


Fig.3 - Depth of surface at rear side of anchor



(a) Mixing with conventional blade *(b) Mixing with newly designed blade*
Fig.4 - Flow velocity distribution during mixing process

Conclusions

By using Particleworks for the simulation, Shiseido could visualize the mixing process and realize an effective mixing blade design. In the production process, it's necessary to take advantage of such simulation technologies for achieving the target quality. For Shiseido today, Particleworks is an essential for creating new value.

Acknowledgements
The author and Prometech Software wish to thank Ms. Tanaka of Shiseido Co., Ltd. for the permission to have an opportunity to introduce their important research projects using Particleworks.

Sunao Tokura, Prometech Software Inc.,

The software Particleworks

Particleworks is a CFD software based on an advanced numerical method known as the Moving Particle Simulation (MPS) method. The mesh-free nature of MPS allows for robust simulation of free-surface flows at high resolutions, saving the need to generate meshes for the fluid domain. Since its first release in 2009, Particleworks has been introduced to a wide range of industries.
Engineers can model and analyze free surface flow, large deformation, and coupled phenomena including fluid, powder and rigid body without complication. Non-Newtonian flow and heat transfer can be combined as well. Preprocessing is straightforward without the need for mesh generation along with direct file import form CAD software, making it easy for users to simulate complex systems.
With the enhanced support for multi-core CPUs and GPUs, Particleworks shows competitive scalability from desktop PCs to large-scale servers. The integrated, easy-to-use interface of Particleworks enhances productivity throughout modeling, simulation and post-processing.

The new release Particleworks5 is focused on extending basic solver capabilities including flexible boundary conditions, improved distance function generation, and automatic simplification of complex shapes. The redesigned user interface, along with accelerated graphics processing, allows for an intuitive user experience. In addition to the new enhancements, coupled simulation with RecurDyn (Function Bay, Inc.) offers a variety of inflow boundaries, and complex motion configurations.

Particleworks is produced by Prometech Software, Inc., Japan and is promoted and supported in Europe by EnginSoft.

For more information:
Massimo Galbiati, EnginSoft
m.galbiati@enginsoft.it

Beyond the barrier of perfection

This article discusses the design optimization of an axial steam turbine (rated power of 160 MW), focusing on maximizing the total-to-total isentropic efficiency of the last three low pressure stages. The turbine, which is designed and produced by Franco Tosi Meccanica SpA, was optimized in collaboration with EnginSoft, by using a modeFRONTIER work flow to explore different designs and completely manage the fluid dynamics simulations. This simulation management involves geometry generation, model pre-processing, solving and post-processing, all achieved through the tools provided by ANSYS. Thanks to a strategic selection of input parameters, output values, targets and project constraints, a wide exploration of the possible parametric space is achieved by using efficient optimization algorithms.

Introduction
While everyone is familiar with the idea of boiling water for cooking or making a coffee, few consider that water has been boiled for nearly everything we do! Thanks to this, every day we can work at our computers, charge our smartphones or relax watching TV. Even if it sounds unusual, it's not too far from the truth. In fact, almost 80% of the electric power we consume comes from power plants that generate electricity from steam.

Hystory of Steam Turbine Technology
The first device to be classified as a steam turbine was the aeolipile, proposed by Greek mathematician Hero of Alexandria in the 1st century. Other steam-driven machines were described in the next centuries; in 1551 by Taqi al-Din in Egypt, in 1629 by Giovanni Branca in Italy and in 1648 by John Wilkins in England. No significant developments occurred until the end of the 19th century when various inventors laid the groundwork for the modern steam turbine. In 1884 Sir Charles Algernon Parsons, a British engineer, recognized the advantage of employing a large number of stages in series, allowing extraction of the thermal energy in the steam in small steps. The invention of Parsons' steam turbine made cheap and plentiful electricity possible and revolutionized marine transport and naval warfare.

After Parsons, a number of other variations of turbines have been developed that work effectively with steam. During the 1880s Gustaf de Laval of Sweden constructed small reaction turbines that turned at about 40000 RPM. From 1889 to 1897 de Laval built many turbines with capacities from about 15 to several hundred horsepower. Auguste Rateau of France first developed multistage impulse turbines during the 1890s. At about the same time, Charles G. Curtis of the United States developed the velocity-compounded impulse stage. One of the founders of the modern theory of steam and gas turbines was Aurel Stodola, a Slovak physicist, engineer and professor at the Swiss Polytechnical Institute in Zurich.

By 1900 the largest steam turbine-generator unit produced 1.2 MW, and 10 years later the capacity of such machines had increased to more than 30 MW. This far exceeded the output of even the largest steam engines, making steam turbines the principal prime movers in central power stations after the first decade of the 20th century. Steam turbines also gained preeminence in large-scale marine applications, first with vessels burning fossil fuels and then with those using nuclear power.

Despite the introduction of many alternative technologies in the intervening 120 years, nowadays it is estimated that more than 80% of the world's electricity is generated using steam turbine systems driving rotary generators. Steam to drive these turbines is raised by burning fossil fuels, mostly coal but also oil and gas (~65%), or by nuclear power (~15%). Less common thermal sources for steam generation are solar power and geothermal energy. Because of its ability to develop tremendous power within a relatively small space, the steam turbine has superseded all other prime movers, except hydraulic turbines, for generating large amounts of electricity and for providing propulsive power for large, high-speed ships. Today, units capable of generating up to 2 GW of power can be mounted on a single shaft.

How Engineers can get a better Steam Turbine Design?
Today, customers demand greater and greater performance. They continuously push to enhance the efficiency of steam turbines aiming to decrease CO_2 emissions from fossil power plants and to increase electrical power output from nuclear power plants. So suppliers are then asked to improve their designs, so steam turbine designers work on getting as much energy as possible out of the steam that is fed in by redesigning the turbine itself. Every day, they are practically asked to give an answer to the following questions: can the turbine be made lighter (so it spins faster) but still strong enough to withstand the heat? Can multiple stages be used to extract energy that would otherwise be wasted? Can heat losses be reduced (by insulating the machine)? What shape should the blades be and at what angle should they be made?
This article will focus on how Franco Tosi Meccanica and EnginSoft tried to give an answer to this last question.
Starting from Parson's concept, after more than a century of development, advances in blading design have contributed to improved steam turbine thermal efficiency. Considering that modern turbines' efficiency can reach values over and above 90%, it is clear that any further improvement is a very challenging task to accomplish.

Theoretical methods and experimental tests are very useful in predicting and verifying the performance of every new design or redesign. However, the classical approach of "trial-and-error" through many experimental tests is very expensive in terms of time and money, while it is also unable to identify how to improve performances exactly.
With the availability of large computer power and efficient numerical algorithm, CFD becomes an essential tool for engineers, enabling a wide variety of complex flow situations to be simulated, reducing the amount of testing required, increasing understanding and accelerating development. As a result of these factors, CFD is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world.
Today the designer has to cope with 2 key challenges to compete in the market: competitive market sales targets and strict energy efficiency regulations. In this complex sales scenario the designer is thus focused every day in "raising the bar", knowing that a few percent increase in performance often makes the difference. This is why a tool must be able to give an accurate, reliable, and automated prediction of the fluid flow behavior in steam turbines to allow Franco Tosi Meccanica to gain a new competitive edge in the market. In this context ANSYS proves to be a high-fidelity CAE tool to match a turbine designers' needs.

Fluid dynamics optimization of a LP Steam Turbine
The last three stages of a low pressure steam turbine is the focus of this study (Figure 1 and Figure 2). The objective is to optimize the three statoric rows by maximizing the total-to-total isentropic efficiency of the device at a given operating condition.
The turbine blades' shape is defined by the position of 5 airfoils in the spanwise direction. A Bezier distribution for both stacking angle (for flow incidence control) and bowing angle (for flow separation control) is used to recreate the blade shape. Another variable is used for the number of blades in the statoric row. See Figure 3 and Table 1 for details.

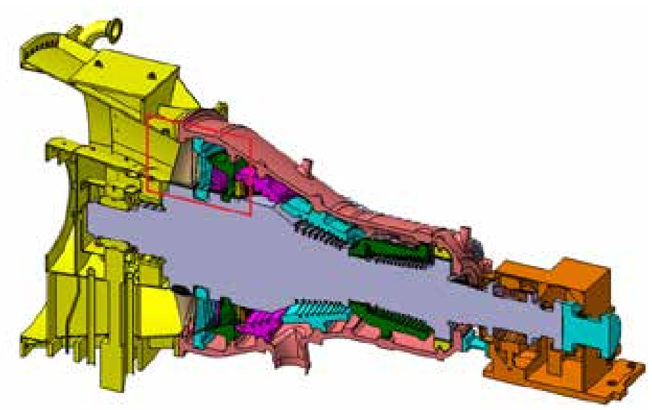


Figure 1 – Multi-stage steam turbine design by Franco Tosi Meccanica – The last three low-pressure stages are higlighted in the red box

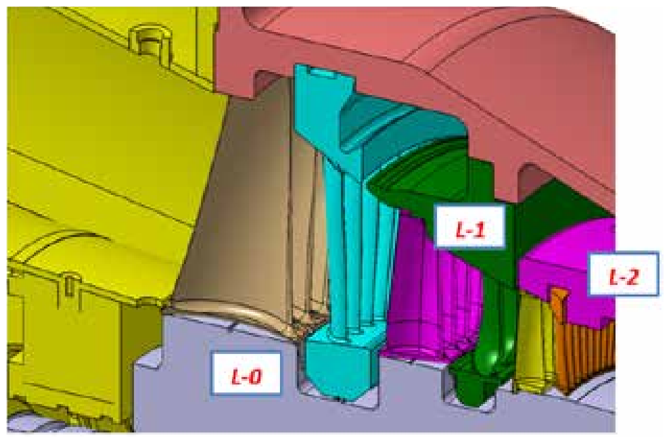


Figure 2 – Multi-stage steam turbine design by Franco Tosi Meccanica – Details of the last three low-pressure stages – Sealings and cavities

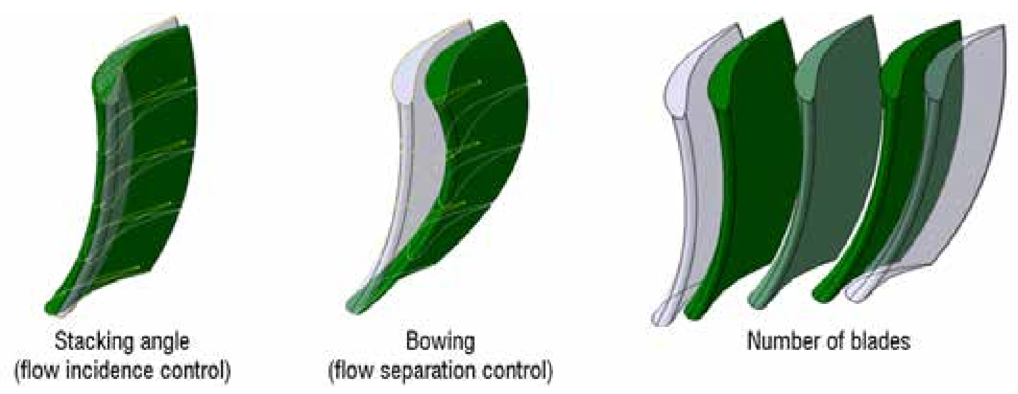


Figure 3 – Blade geometry – Description of parameters

A parametric model is generated with this characteristics in ANSYS DesignModeler. Starting from here, through ANSYS BladeEditor features, it is possible to extract the computational domain for a single passage. In fact, the first assumption made here is to consider just a single passage instead of a full wheel (Figure 4). The rotating speed compared to the mean stream velocity is such that a mixing plane approach is well suited. Consequently we obtain a huge advantage in terms of reduction in the computational costs, both for the size of the CFD model and for the steady state approach with mixing plane.

Another assumption made is the simplification of the actual geometry of the steam turbine: sealings, cavities, and rotor are neglected in the computational domain. This is required to make a completely automatic optimization workflow possible, in particular for the computational grid generation of ANSYS TurboGrid (Figure 5). A grid independency study is performed in order to apply the best compromise between speed and accuracy. The result is a very fast and robust procedure able to achieve a high quality mesh and a well-defined boundary layer treatment for every configuration which is identified by a unique parameter set, or design point.

The result of the optimization is the geometry for three statoric blades. This generation depends on the simplified ("ideal") layout of the flowpath considered here. Then, after the optimization campaign, a final comparison between the "ideal" flow path and the actual one is performed in order to measure the losses due to sealings and cavities.

Another assumption made is to consider each stage independently from each other in the optimization. In this way the three stages, namely L-0 (the last one before the diffuser), L-1, and L-2, are treated separately in three different optimization stages. Boundary conditions for each CFD model are obtained combining 1D data supplied by Franco Tosi Meccanica and a preliminary analysis of the single passage, three-stage turbine performed by EnginSoft. An additional analysis of the single passage, three-stage turbine is performed after each optimization stage. The baseline geometry of the statoric blade is replaced by the geometry that represent the result of the current optimization stage. In this way we want to verify if the new layout of the turbine performs better than the baseline configuration. Once this is assured, the optimization carries on to the next stage.

ANSYS CFX is used to set up and solve the CFD analyses. The fluid flow is considered in steady, compressible, and turbulent conditions. The advection term is resolved with "High Resolution" scheme (bounded 2nd order accuracy). The RANS 2 equations Shear Stress Transport (SST) is chosen for turbulence model. The IAPWS library is employed to characterize the steam as a real fluid. The liquid-vapor phase transition is considered in equilibrium conditions. The Stage (or mixing plane) approach is considered for multiple frame of reference (MFR). The numerical setup has been optimized in order to achieve a good level of convergence in less than 100 iterations. In this way, each design point selected in the optimization campaign took just about 25 minutes to be estimated (from the selection of the parameter set to the output of the post-processing procedure).

		Minimum	Maximum	Step
STACKING ANGLE	Rotation angle of the profile sections around their centroid ($\Delta\theta$)[1]	-5.0°	+5.0°	0.2°
	Maximum difference between 2 consecutive sections		±5°	
BOWING ANGLE	Rotation angle of the profile sections around turbine axis ($\Delta\beta$)[1]	-3.0°	+2.0°	0.1°
	Maximum difference between 2 consecutive sections		1.5°	
NUMBER OF BLADES	Stage L-2	72	76	2
	Stage L-1	64	72	2
	Stage L-1	146	154	2

Table 1 – Blade geometry – Description of parameters

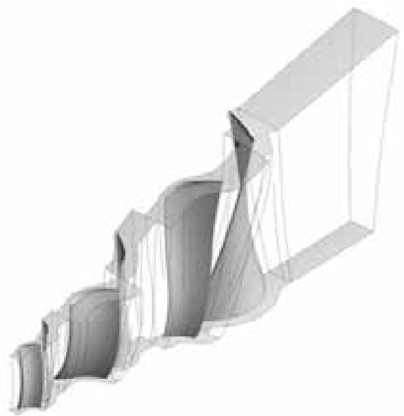


Figure 4 – Three stage, single passage – CFD model

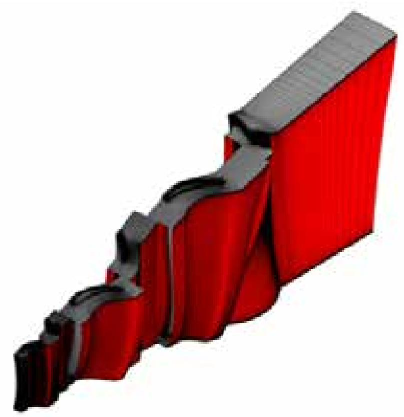


Figure 5 – Three stage, single passage – Computational grid

The results collected by the post-processing procedure are useful in understanding if the new design performs better with respect to the baseline configuration. The objective is to maximize the total-to-total isentropic efficiency:

$$\eta_{TOT-TOT} = \frac{h_{0\,TOT} - h_{1\,TOT}}{(h_0 - h_{1\,ISO}) + \frac{1}{2}(v_0{}^2 - v_1{}^2)}$$

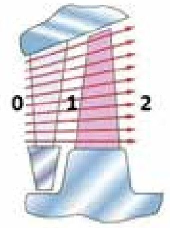


Figure 6 – Reference for post-processing

Several operating constraints have to be satisfied in order to guarantee the feasibility of a design point. In particular, such constraints are (see Figure 6 for reference). The choice of input parameters, output values, targets and project constraints greatly define an optimization process. Building from this point, several techniques can be selected defining what kind of optimization method to apply to an engineering problem is appropriate, especially in terms of time and cost.

			Minimum	Maximum
EXPANSION RATIO		$\frac{p_0}{p_2}$	-5%	+5%
REACTION RATIO		$\frac{h_1 - h_2}{h_0 - h_2}$	-10%	+10%
VAPOUR FRACTION		L-0	90%	
$\frac{\dot{m}_{vapor}}{\dot{m}_{total}}$		L-1	94%	
		L-2	100%	

Table 2 – Operating constraints (with respect to the baseline configuration)

Traditional engineering based on "trial-and-error" and was widely used in the past when automatic optimization tools were not available. Starting from a baseline configuration, the design is perturbed in order to get a new design point (hopefully with improved performance). The aspects of such perturbation is selected and applied by the designer, basically according to his experience. This process is repeated iteratively until the desired performance target is reached. It is clear that this approach cannot be completely automated, because decisions are made by the designer, and it is a time consuming and costly method. A schematic representation of this approach is shown in Figure 7.

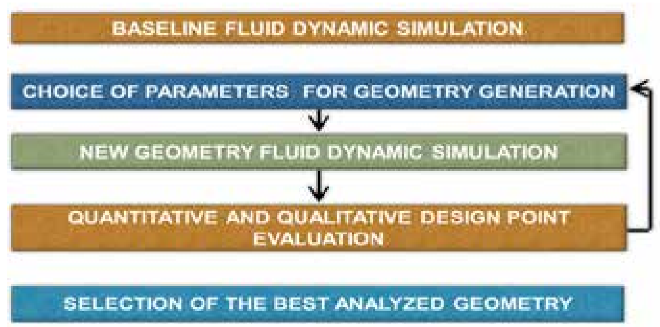


Figure 7 – Optimization methods – "Trial-and-error" approach

The evolution of this concept lead to the birth of modern optimization approaches, where algorithms took the place of the designer in the selection of the new design points to be evaluated. By means of efficient optimization algorithms, a wide and intelligent exploration of the parametric space can be performed in a completely automated way in much less time compared to traditional "trial-and-error" approach. Two different approaches are available:

- direct optimization: after a first exploration of the parameters' space through a DoE (Design of Experiment), each of the design points is explicitly evaluated, aiming to reach a desired target. See Figure 8;

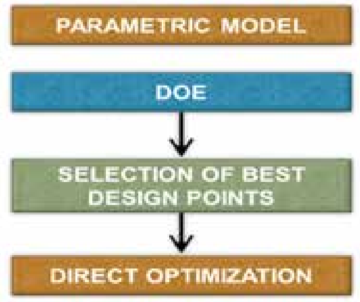


Figure 8 – Optimization methods – Direct optimization approach

- virtual optimization: after a first exploration of the parameters' space through a DoE, a response surface is generated in order to have a continuous representation of this space. In turn, this response surface is explored with a large number of virtual design points (not explicitly simulated) in order to find good candidates. Then, these good candidates are explicitly evaluated, enriching the database on which the response surface can be redefined. This process is repeated iteratively until the possible inaccuracy of the response surface goes below a certain threshold. See Figure 9.

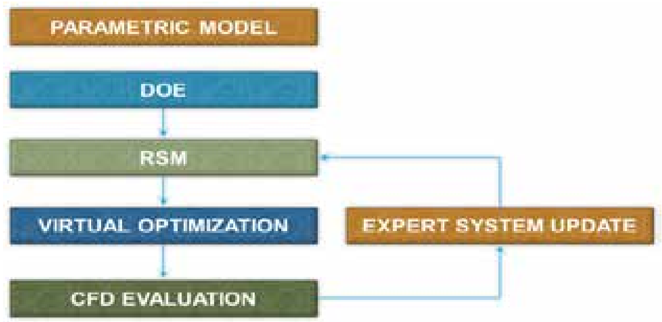


Figure 9 – Optimization methods – Virtual optimization approach

For direct or virtual optimization approaches we need an optimization software that is able to control the workflow that automatically manages both the exploration of different design points and the management of the fluid dynamics simulations, such as geometry generation, model pre-processing, solution and post-processing by means of the CAE tools provided by ANSYS. The software employed for this purpose is ESTECO's modeFRONTIER.

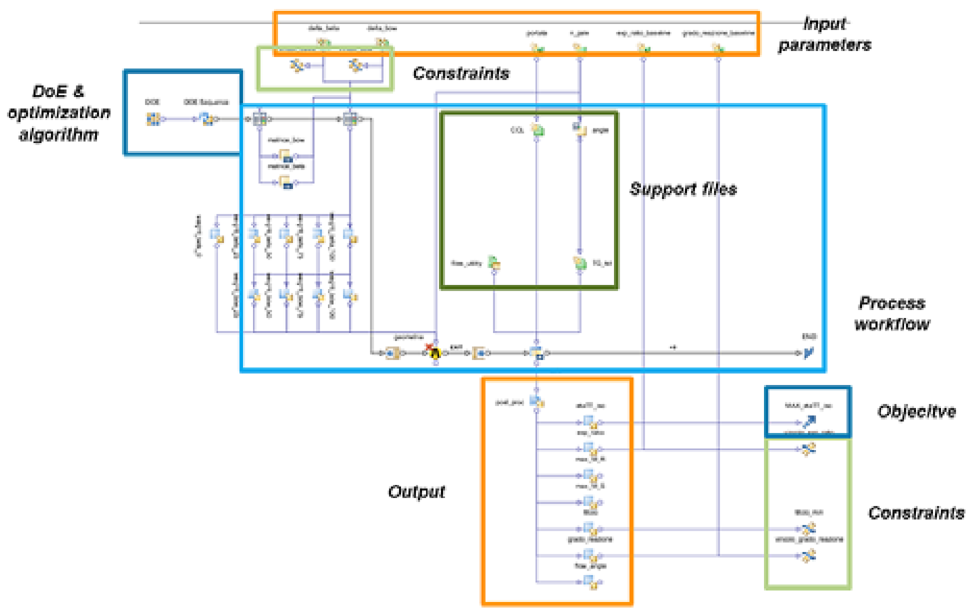


Figure 10 – ESTECO modeFRONTIER – Optimization worklow

Stage	Optimization approach	Optimization algorithm	Performed by	Number of explicit design points evaluated	Human time consumed
L-0	"Trial-and-error"	N.A.	Franco Tosi Meccanica	~ 100	~ 1 month
L-1	Virtual	MOGA-II + RSM	EnginSoft	~ 1000	~ 1 month
L-2	Direct	MOGA-II	EnginSoft	~ 5000	~ 1 month

Table 3 – Optimization strategy

Stage	Single stage improvement	Improvement of the three-stage turbine
L-0	+1.0%	+0.4%
L-1	+1.0%	+0.6%
L-2	+0.9%	+0.7%

Table 4 – Results of the optimization campaign

modeFRONTIER's optimization process could be split into 3 steps:

- the first step includes the creation of a logic workflow in order to graphically formulate the engineering design problem at hand, i.e. how the simulations have to be performed and in which order. The "how" implies the choice of values/measures to be used and generated (inputs and outputs), the definition of the optimization objectives and the configuration of the most adequate algorithms for design space exploration and optimization. A representation of the optimization workflow defined for this study is shown in Figure 10;
- the second step consists in the evaluation of designs, as defined by the workflow. The evaluation, or "run", can be monitored in real-time by means of charts and graphs, and direct access to log and process files;
- The final step is the assessment and visualization of results. The available tools allow understanding of a problem's important parameters on the basis of the design space exploration, reducing the number of significant parameters considered making the optimization more efficient, re-arranging data in a comprehensible manner and extracting a clear meaning in order to make informed decisions. Specific analysis tools help convey relevant insights on the interaction effects and visualize optimization trends. The RSM (Response Surface Models) tool allows for the training, comparison and validation of meta-models, speeding up the entire optimization process.

In this study, all the three different approaches have been used, as summarized in Table 3.

In the classical "trial-and-error" approach, all the design points (almost 100) have been explicitly evaluated by Franco Tosi Meccanica designers on their local workstations. This optimization stage took about one month to be accomplished.

For the optimization stages in which L-1 and L-2 were studied EnginSoft adopted an automatic optimization procedure by means of modeFRONTIER™. MOGA-II (Multi-Objective Genetic Algorithm) is the optimization algorithm selected for this campaign. The optimization stage for L-1 required the evaluation of about 1000 design points, while for L-2 the CFD calculations performed were about 5000. All these computations were performed on the EnginSoft cluster. A concurrent design points strategy has been adopted to reduce the total wall clock time, so both the optimization stages took about one month to be completed.

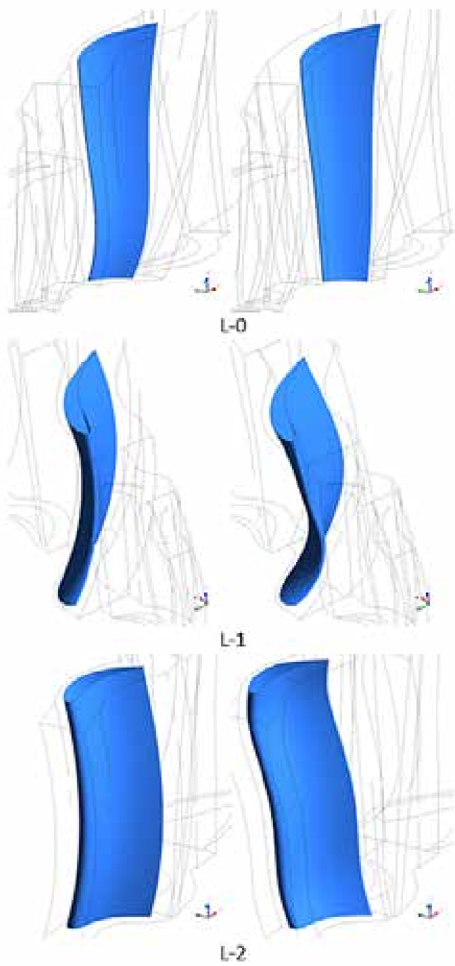


Figure 11 – Comparison between baseline (left) and optimized (right) design of the statoric blades – Blade geometry

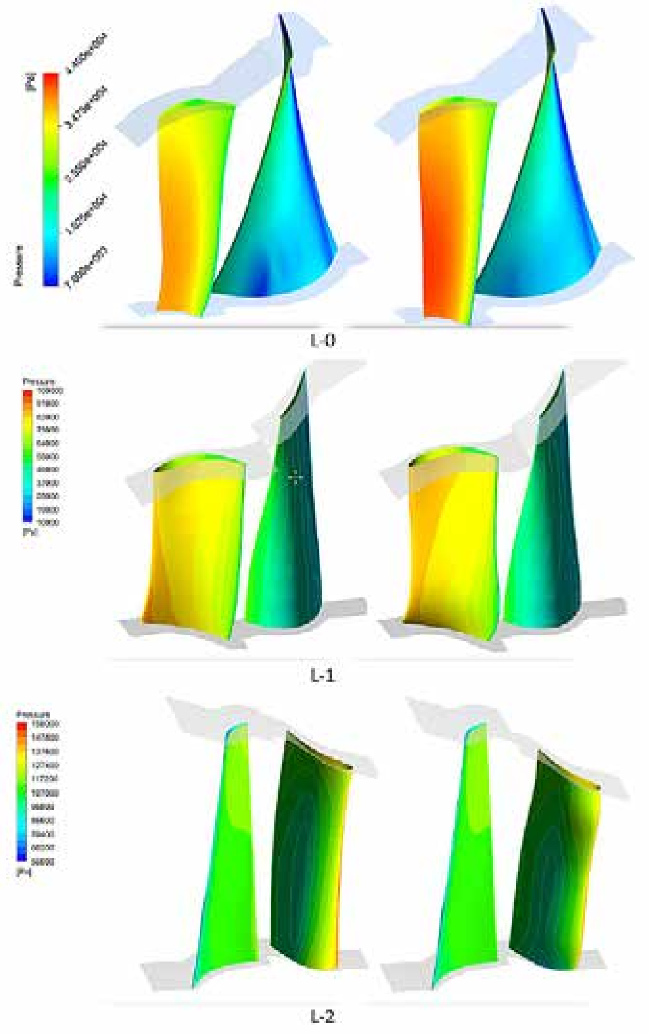


Figure 12 – Comparison between baseline (left) and optimized (right) design of the statoric blades – Pressure field on blades

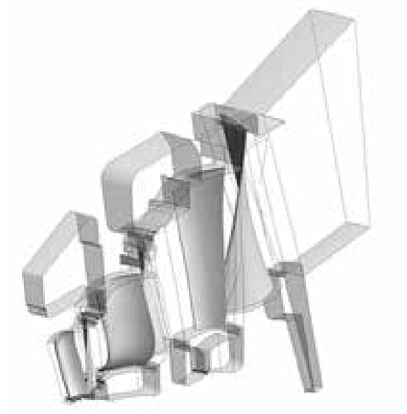


Figure 13 – Three stage, single passage – CFD model with actual sealings and cavities

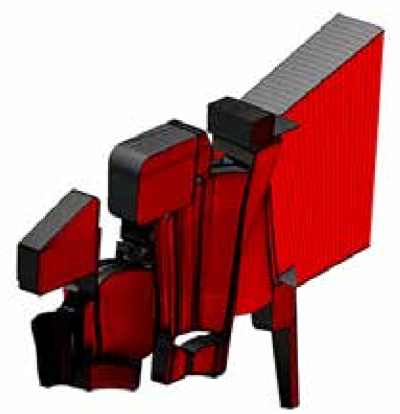


Figure 14 – Three stage, single passage – Computational grid with actual sealings and cavities

The results of the optimization campaign is summarized in Table 4. The second column represents the improvement of the total-to-total isentropic efficiency evaluated on the respective turbine stage. In other words, these are the results of the three optimization stages. The last column represent the results of the additional analyses of the single passage, three-stage turbine that are performed after each optimization stage. As mentioned before, in these analyses the baseline geometry of the statoric blade is replaced by the geometry that represent the result of the respective optimization stage.

Even if the optimization stages are independent from each other, it is clear that a good overall trend is achieved. It is good to highlight that the efficiency of the baseline design is quite high (above 90%). From this point of view, such results are very remarkable.

The last phase of the study was the evaluation of the single passage, last three-stage steam turbine with actual flowpath, i.e. including sealings and cavities (see Figure 13 and Figure 14). A comparison between baseline and optimized geometries of the statoric rows has been performed. In real conditions, the improvement of the total-to-total isentropic efficiency is about 0.5%.

Conclusion

The object of the study was to optimize the last three low pressure stages of a steam turbine. A global optimization has been performed on such system. Acting on the geometry of the statoric blades, the purpose was to find the optimal performance in terms of isentropic total-to-total efficiency.

The optimization strategy adopted here was defined on 3 different stages with different approaches:

- "Trial-and-error" for L-0
- Virtual optimization (by means of response surface methods) for L-1
- Direct optimization for L-2

Once the optimal design for each stage has been selected, it has been fit into the three stages steam turbine in order to confirm the improvement of the system.

After the 3 optimization stages, a final verification of the steam turbine was performed, taking into account the real flow path with sealings and cavities. In this complex scenario, the isentropic total-to-total efficiency gain is about 0.5%.

It is worth noting that, as summarized in Table 3, each optimization stage requires the same amount of human engineering time (~1 month). This time includes all the CPU time and man hours involved. In the last optimization stage, the engineer's time is almost close to zero while the simulations are run automatically. Given that each design takes about 25 minutes to run on the

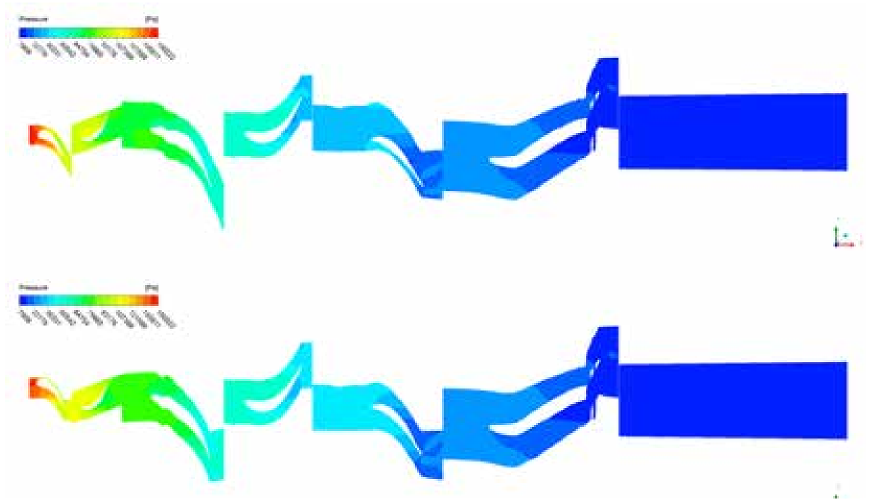


Figure 15 – Three stage, single passage – CFD model with actual sealings and cavities – Static pressure field comparison @ 50% span – Baseline (above) and optimized (below)

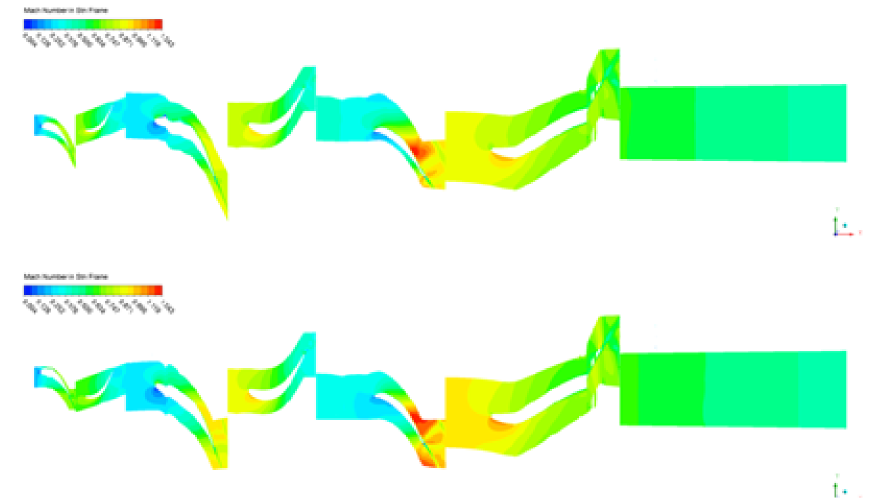


Figure 16 – Three stage, single passage – CFD model with actual sealings and cavities – Mach number field comparison @ 50% span – Baseline (above) and optimized (below)

EnginSoft cluster, and 3 concurrent design points were running simultaneously during the optimization stage, the total CPU time is about 1 month.
Now, in a typical medium enterprise, where HPC computing clusters of over a hundred CPUs are now quite common, it is clear that such times could be dramatically reduced. For example, for a medium installation of 256 cores, the same job could be performed in just 1 week! This is a remarkable result, especially if we think about the time that can be saved in the development of a large machine like the one considered in this study.

Emanuel Pesatori, Head of product development
Andrea Radaelli, Turbomachinery specialist
Giorgio Turozzi, Head of Engineering
Giovanni Zamboni, Turbomachinery specialist
Franco Tosi Meccanica S.p.A., Legnano, Italy

Lorenzo Bucchieri, CFD Manager
Alessandro Marini, Mattia Olivero
CFD Project Managers
EnginSoft, Bergamo, Italy

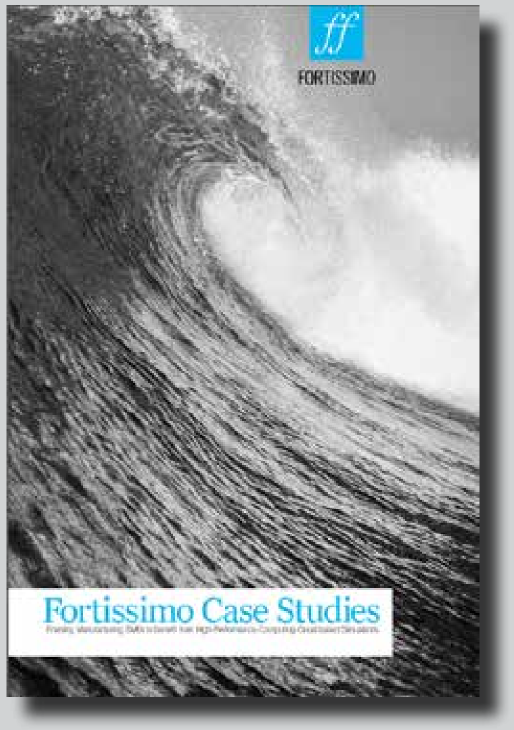
For more information:
Lorenzo Bucchieri, EnginSoft

The Fortissimo Project

Fortissimo is a collaborative project that enables European SMEs to be more competitive globally through the use of simulation services running on a High Performance Computing cloud infrastructure. The project is coordinated by the University of Edinburgh and involves 123 partners including Manufacturing Companies, Application Developers, Domain Experts, IT Solution Providers and HPC Cloud Service Providers from 14 countries. These partners are engaged in 53 experiments (case studies) where business relevant simulations of industrial processes are implemented and evaluated. The project is funded by the European Commission within the 7th Framework Programme and is part of the I4MS Initiative.

Fortissimo Booklet and Case Studies

The Fortissimo booklet contains a collection of 15 case studies from the first wave of initial experiments. They demonstrate the wide verity of HPC-cloud solutions and the impact these solutions had on the business of manufacturing SMEs as Fortissimo experiment partners. You can download your free copy of the Fortissimo booklet at: http://www.fortissimo-project.eu/
For more information on Fortissimo Project:
Lorenzo Bucchieri, EnginSoft - l.bucchieri@enginsoft.it



Challenges when modeling the complex vessels and machinery used in the Oil & Gas Industry

Gemma Church, a UK-based freelance writer specializing in business, science and technology for Scientific Computing World, investigated with Livio Furlan, Chief Technical Officer for EnginSoft's Structural and Oil&Gas Competence Centre, how simulation software models the equipment used in the Oil &Gas Industry.

The December 2015/January 2016 issue of the Scientific Computing World Magazine (http://www.scientific-computing.com/features/feature.php?feature_id=492) highlights the interview conducted, with the complete interview reported below.
It has to be noted that this interview was inspired by articles written by Livio Furlan, published in the previous editions of the EnginSoft Simulation Based Engineering & Sciences Newsletter Year 8, n. 4, Winter 2011 (FSO and Shutter Tanker Tandem Configuration Hydrodynamic Analysis) and in Newsletter Year 10, n. 2, Summer 2013 (QCDC Connector Design on a FPSO Unit), which also features in the Scientific Computing World article.

The two case studies cover two very different simulations - how do you cope with the diverse nature of machinery that the oil and gas industry uses?
The diverse nature of oil and gas machinery emanates from the many different contexts in which machines are used, such as drilling, production, subsea or downstream processes. This in turn governs the varied design scopes required. So to cope with the different characteristics of any machine it is essential to know the background detail. For instance, what is a machine's function in relation to its requisite operating needs and phases (e.g. temporary, long-term) as well as any safety requirements for both human life and pollution?
Additionally, operators, environmental, design and simulation engineers must also be familiar with any State and Federal/Regional regulations governing the management of hazardous and nonhazardous operations/activities in order to establish how to meet the real scenario in which any machinery has to operate with unique characteristics and functionalities. Only by intelligent analysis can any machinery be successfully designed with a near certainty that it can operate according and effectively in the environment for which it has been built.

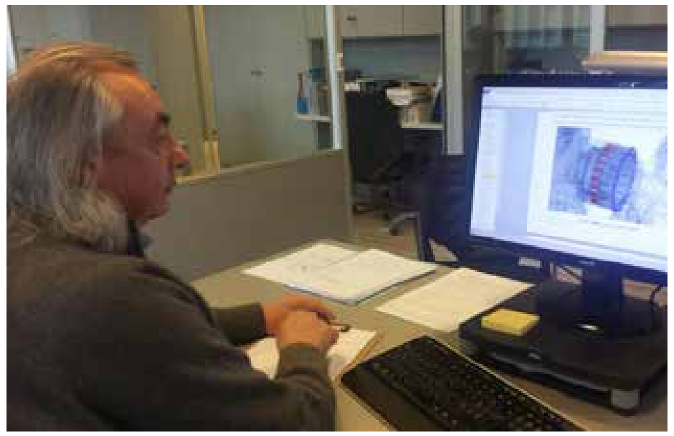


Figure 1 - Eng. Livio Furlan, Chief Technical Officer of EnginSoft's Structural and Oil&Gas Competence Centre

Figure 2 - Rolling-up of Large Auxiliary Buoyancy Tanks for Jacket free-floating

And how do you simulate for the diverse range of conditions that these machines, vessels and components may be faced with?
The first step of simulation, in whatever field it is applied, is to understand the real conditions of the phenomena to be simulated. This creates a reliable, base engineering prototype/model of reality and of any related responses. Available and relevant scientific software can provide very useful results in predicting expected phenomenon that helps drive consequential choices towards a safer, optimal or most fit solution.

In the context of floating vessels, it is now possible to assess their behaviors under a wide range of environmental conditions; taking into account waves, wind, and/or current as deterministic and/or stochastic parameters. This means that levels of confidence in results reliability allows us to proceed with the design of systems and components starting with the certainty that any final product/ machine will be fit for purpose because their virtual simulation is based on a comprehensive and correct representation of real working conditions. This is therefore relevant to the study of mooring lines behavior, related fatigue assessments or even designing a multi-bore connector, as long as behind the software/ computer someone has set up the original problem in a correct and proper way.

Are there any further challenges and limitations for your software when dealing with the oil and gas industry?
If correctly 'driven' or 'customized', then software can enormously reduce the distance between reality and any simulation. Let's take deep water activity, probably one of the most crucial and demanding frontiers at present, offering a unique opportunity to add significant volume to the world's proven oil and gas reserves. The real challenge here is not necessarily the software chosen to simulate the phenomenon, but making sure that the final design of technically and economically safe machinery is sufficient to operate under critical conditions.
So in environments and conditions such as these, it's back to the intelligent analysis of detail that will give us 'as best as we can get' to reality so that efficiency and risk are assessed upfront, during the numerical design phase. Conventional trial-and-error procedures cannot apply to these critical environments, albeit shop tests are applicable at the end as means to confirm that the design was correct. So a simulation based approach is the only one that can effectively and efficiently determine the best economic design of any machinery/components as well as evaluating different what-if scenarios to ensure required safety and robustness.

How do you address these challenges?
They have to be addressed with constant research, application, diligence, humility as well as enthusiasm and creativity. The right mixture of these components can generate success. It's always people that get things done and when you assemble the correct teams of experts who can undertake a real comprehension of the scenarios and environments where any 'design' is expected, then you have a high probability in identifying/detecting all the critical conditions and the related combinations to taken into account so that challenges are obviated. With this in mind, in depth studies of similar cases are always a good place to start in order to gain a good degree of understanding of the issues involved. And last but not least, harking back to teamwork, a good feeling between passionate and relevant people believing in the same project has seen us break through barriers that others thought insurmountable.

For more information:
Livio Furlan, EnginSoft - l.furlan@enginsoft.it

Figure 3 - Finite Element Model of a Well Head Safety Valve - Detail

Figure 4 - Jacket Launching

CFD Characterization of the Ventricular Assist Device HeartAssist 5® Through a Sliding Mesh Approach

A valid bridge to heart transplant
Cardiovascular diseases are recognized as the main cause of death worldwide, primarily related to heart failure. Owing to insufficient amount of eligible organs for heart transplant annually, various mechanical circulatory support devices as ventricular assist devices (VAD) and total artificial hearts have been developed and introduced in the market. VAD utilization as a mean of stabilizing congestive heart failure patients or as a bridge-to-transplant has increased dramatically over the past few years. In particular, rotary VADs offer the advantages of smaller dimension and simpler structures with respect to pulsatile VADs; however the continuous high-speed rotating blood flow patterns generated are a potential risk factor for adverse events, including thrombus formation, thromboembolic complications and device malfunction. Pump thrombosis is one of the main causes for device malfunction, and patients are exposed to the risk of sudden death or the risks involved in complex device replacement surgery.
In this work, computational fluid dynamics (CFD) simulations were performed to mimic the realistic operative conditions of the VAD HeartAssist 5® (HA5, ReliantHeart Inc., USA), reported in Fig. 1. CFD analysis can be exploited to predict blood flow streamlines passing through these devices: thrombus formation arises from the combined effect of elevated shear stress levels and recirculating flow patterns in specific regions within the device.

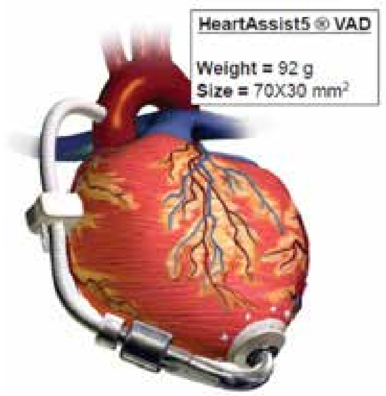


Fig. 1 – Heart Assist5 ®, left ventricular assist device (LVAD)

The advantage of using a sliding mesh technique in ANSYS Fluent
The CAD model of the device (Fig. 2) was provided by the manufacturer and optimized with Gambit 2.4.6 (ANSYS Inc., Canonsburg, PA, USA). The HeartAssist® LVAD has a weight of 92 g and the internal fluid domain was obtained subtracting the CAD model from a cylindrical conduct. The VAD components (Fig. 2) include the inlet flow-straightener, the impeller (rotor) and the diffuser; these parts are connected through a rear hub, connecting the stationary flow-straightener and the impeller, and a front hub, connecting the impeller and the stationary distal diffuser.
Subsequently, the entire domain was discretized, with the Meshing module of ANSYS Workbench v15.0, into 2.5 million of tetrahedrons prescribing a characteristic mean size of 0.3mm. Exploiting the capabilities of the ANSYS CFD solver Fluent, a sliding mesh technique was adopted to simulate the fluid-structure transient rotating behavior of the device (Fig. 2): the rotor-stator interaction can be modeled by allowing the mesh associated with the rotor impeller to rotate relative to the stationary mesh associated with the straightener and the diffuser. Specifically, a rotational speed (ω) was set to the rotor part and two mesh interfaces were defined between the moving (rotor) and stationary (straightener, diffuser) domains. In order to simulate the clinical operating condition of the device, the impeller spins at 9,000 rpm clockwise, generating a corresponding cardiac output of 4.0 L/min.

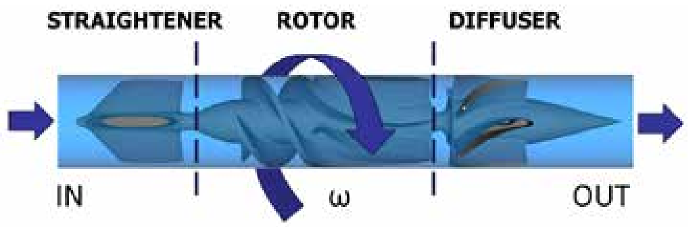


Fig. 2 – HeartAssist5 ® CAD model

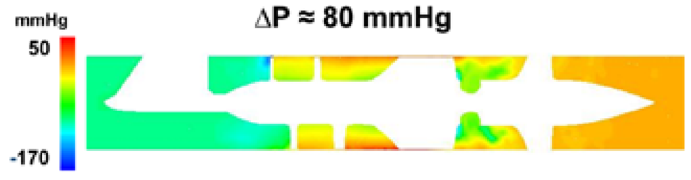


Fig. 3 – Static pressure (mmHg) contour on a longitudinal plane to highlight the pressure head of the device

The sliding mesh interface was assigned to the impeller, which external wall was modeled as "moving relative to adjacent zone", with an opposite rotating direction. The remaining surfaces were defined as stationary walls with no slip condition. A mass flow rate of 0.07 kg/s, corresponding to a physiological Cardiac Output of 4 L/min, was set at the inlet boundary surface. A zero pressure was set at the outlet boundary surface. Blood was modeled as a Newtonian incompressible fluid with viscosity (μ) of 0.0035 kg/m-s and density (ρ) of 1,081 kg/m3. The simulation was run in ANSYS Fluent (ANSYS Inc., Canonsburg, PA, USA), exploiting the Navier-Stokes governing equations, on an Intel Xeon (2.93 GHz) workstation with 24 processors.

CFD testing of HeartAssist® LVAD performance under realistic working conditions

The sliding mesh numerical approach proved able to replicate the operating conditions of the HA5 LVAD, within a physiological regime. We observed a pressure head of the device of approximately 80 mmHg capable of easing the load of the left ventricle (Fig. 3). High pressure-loss regions suggested possible optimization of the LVAD design, as depicted for instance near the distal part of the rotor.

Moreover, ANSYS CFD-Post was used to obtain a qualitative visualization of the velocity field by means of streamlines calculation (Fig. 4, upper panels). Specifically, the interface region between the straightener and the rotor (I) depicted a laminar rotational flow, whereas the one between the rotor and the diffuser (II) highlighted a residual rotational tendency of the flow field.

Velocity magnitude 3D contour maps (Fig. 4, lower panels) were extracted at different locations of the fluid domain. On the one hand, the rotor part was characterized by the highest values, close to 6 m/s, with a laminar distribution of blood flow; whereas, on the other hand, the diffuser was characterized by lower velocities but residual flow helicity. The outflow tract was closer to physiological values of 1 m/s and was characterized by the reduction of helical flows. Furthermore, wall shear stress maps were computed to highlight locations that may be associated to the highest blood damage due to elevated shear stresses (Fig. 5). The highest peak values were noticed in the proximal part of the rotor and evidently associated with both the impact of blood flow hitting the device as well as the high-spinning rate of the rotor.

In conclusion, high-end numerical CFD simulations allowed to evaluate the HA5 LVAD hemodynamics under realistic working conditions, highlighting locations of sub-optimal performances of the device. Hence, best exploiting the potential of ANSYS Fluent, the proposed approach may be pivotal in order to deepen VAD fluid dynamics, to estimate its hemolytic potential and optimize its design, potentially leading to enhanced safety and efficacy of VADs for long-term destination therapy.

Biomechanics Group www.biomech.polimi.it
Department of Electronics, Information and Bioengineering
Politecnico di Milano, Milano.

This work was accomplished in collaboration with the Biofluids Research Group (Stony Brook University, Stony Book, NY, USA).

For more information on Biomedical CAE applications:
Alessandra Pelosi, EnginSoft
a.pelosi@enginsoft.it

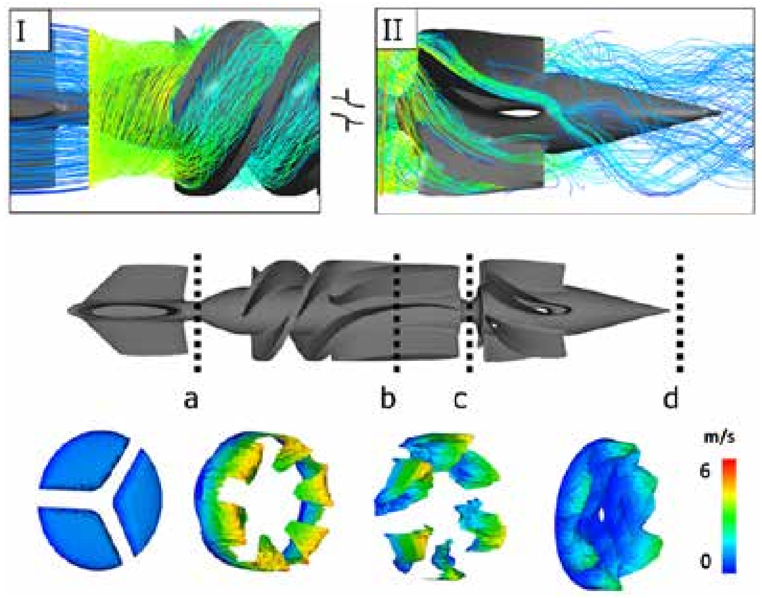


Fig. 4 – Upper panels: streamlines calculation to highlight the velocity field before (I) and after (II) the rotor. Lower panels: velocity magnitude (m/s) 3D contours on different cross-sectional planes (a – d)

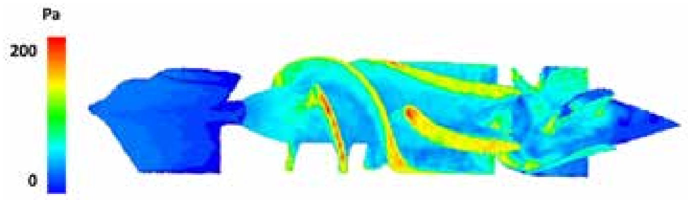


Fig. 5 – Wall shear stress computation (Pa) to highlight high-risk locations of the device prone to blood damage

Propeller design by means of multiobjective optimization: **CP propeller test case**

UNIVERSITÀ DEGLI STUDI DI GENOVA

1. Introduction

Propeller design has evolved significantly in the last few years, with the introduction of numerical methods which can provide an ever improving assessment of propeller characteristics, considering propeller non stationary functioning and cavitating behavior, not only in correspondence to the usual design conditions, but also to off-design conditions. This assessment has become widely adopted, with numerical methods being able to predict propeller characteristic curves (and cavitating behavior) in correspondence to a wide range of advance coefficients. Modern propeller requirements involve many different characteristics, not limiting only to maximum efficiency, but considering also propeller cavitating behavior and, more and more, its side effects, in terms of radiated noise and pressure pulses. This is evident with the ever-increasing demand for improvement of comfort onboard and discussions about radiated noise problems, especially in proximity of protected areas.

In recent years, the interest towards the problem of radiated noise has led the EU to fund some cooperative projects, i.e. SILENV, AQUO and SONIC, whose activities were concluded at the end of 2015. In the context of the first project, UNIGE was involved in the analysis of different ways to reduce underwater radiated noise; since the marine propeller, when cavitating, may become the most significant noise source of a ship, a large part of the work was devoted to the application of a design procedure in which in-house panel codes are coupled with the modeFRONTIER, a multiobjective optimization software. This procedure had already been proposed in (Gaggero, S. and Brizzolara, S. 2009. Parametric CFD Optimization of Fast Marine Propellers FAST 2009), however in the context of this project it was also possible to test it against a real case study and, most important, to validate the results by means of an experimental campaign carried out at Genoa University Cavitation Tunnel.

In particular, the test case is represented by a CP propeller originally installed onboard a RORO-Ferry ship; the peculiar propulsion arrangement of the ship is characterized by having an almost constant revolution rate of the propeller, achieving different operational speeds by means of propeller pitch reductions. In the optimization loop, consequently, two very different working conditions were considered, i.e. the usual design condition at maximum speed and a very reduced speed, obtained at constant RPM and reduced pitch. The two conditions are characterized by very different cavitating behavior, with presence of back sheet cavitation and tip vortex at maximum speed condition and of vortex from sheet face and sheet face cavitation at reduced pitch condition. The latter, in particular, resulted in noise and vibration problems in correspondence to this operating condition, as remarked by the shipowner. As a consequence, the scope of the optimization activity was a new design with the main attention given to the reduced pitch, with the aim of reducing propeller radiated noise, trying contemporarily to keep cavitation extent as low as possible at maximum speed and maintaining propeller hydrodynamic characteristics (with particular attention to propeller efficiency). The optimization strategy is briefly described in section 2, while in section 3 the actual optimization activity is presented. Finally, in section 4 the results of the experimental campaign carried out in order to validate the results are reported.

2. Theoretical background and optimization setup

Traditional propeller design methods are based on lifting line and lifting surface codes; their utilization has been established for a long time and form, even nowadays, the most used tool for propeller designers. Nevertheless, these methods cannot be used directly for a propeller, which needs to be designed for very different working conditions, as the one considered in this work. From this point of view, coupling of panel

codes with optimization algorithm can represent an efficient alternative to the classical approach for the designer, as presented in (Gaggero et al. 2009).
As it is well known, panel codes are usually adopted as a propeller analysis tool, and not for design purposes. However, their coupling with multiobjective optimization algorithms allow their use for design purposes. The main advantages of this approach are that panel codes are capable of capturing better propeller performances, allowing to include also limited and local variations, which could be hardly considered with traditional tools. Moreover, panel codes may allow to consider (at least with a level of accuracy which allows a comparative analysis of different geometries) also off-design conditions, like those related to the reduced pitch proposed in present work; finally, the multiobjective algorithm may allow to consider contemporarily very different working conditions, optimizing them contemporarily.
An accurate description of the panel code utilized in this work may be found in Gaggero et al (2009), and is here omitted for the sake of brevity. In order to apply systematically the panel code inside the optimization loop, a robust parametric representation of the propeller geometry (Gaggero et Al. 2009, Brizzolara, S., Gaggero, S. and Grasso, A. 2009. Parametric Optimization of Open and Ducted Propellers, Propeller/ Shafting symposium 2009) is needed.

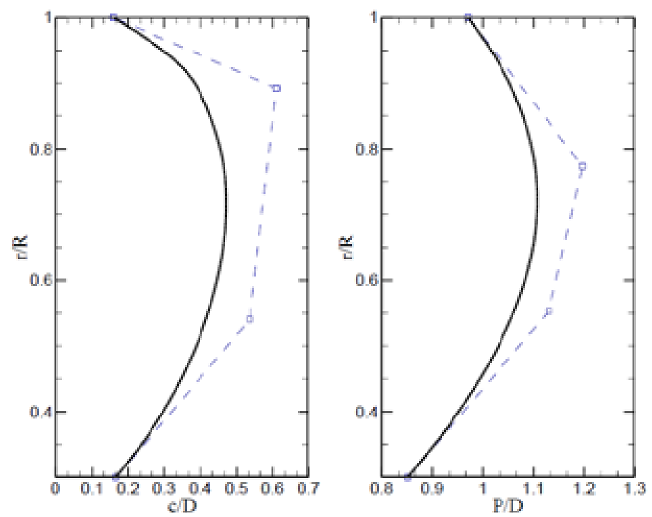


Fig. 1 - B-Spline representation of radial distributions of chord and pitch. [Bertetta, D., Brizzolara, S., Gaggero, S., Viviani, M., Savio, L. 2012 "CP propeller cavitation and noise optimization at different pitches with panel code and validation by cavitation tunnel measurements", Ocean Engineering 53 (2012)]

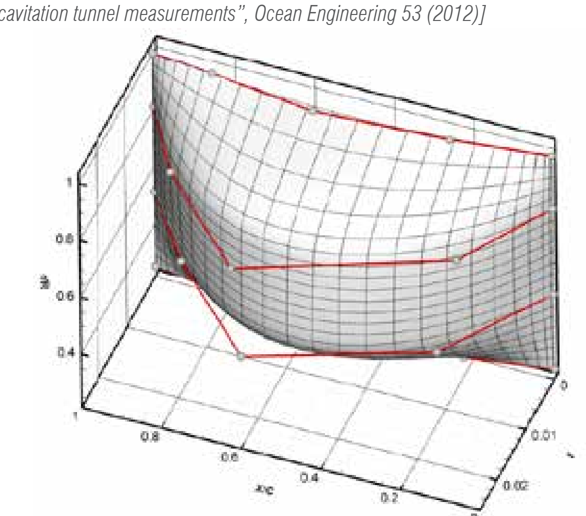


Fig. 2 - Parametric representation of propeller nondimensional mean surface [Bertetta et al 2012]

The classical propeller design table is, inherently, a parametric description of the geometry itself. All the main dimensions that defines propeller geometry, like pitch, camber and chord distribution along the radius, represent main parameters that can easily be fitted with B-Spline parametric curves, whose control points turn into the free variables of the optimization procedure, as in Figure 1.
For what regard the profile shape, instead of adopting standard NACA or Eppler types, with the same parametric approach it is possible to describe only with few control points thickness and camber distributions along the chord for a certain number of radial sections (or, more consistently, to adopt a B-Surface representation of the mean non-dimensional propeller surface) and include also profiles in the optimization routine (Figure 2).
This was actually the approach utilized in this test case. Once a parametric description of the propeller has been obtained, an optimization loop has been built into the modeFRONTIER environment, and optimization has been carried out by means of the MOGA-II genetic algorithm. At each step of the loop (i.e. for each new generated solution), the potential code is used to evaluate the hydrodynamic characteristics of the propeller (in terms of thrust, torque, efficiency and cavity area/volume).
A fully unsteady calculation of propeller behaviour, although carried out with a panel method, would be excessively time expensive to be included in the optimization loop. As a consequence, as presented in Gaggero (2009), unsteady performances are approximated, with a quasi-steady approach, as the mean (or the sum) of the steady performances evaluated in "N" angular wake sectors, whose mean flow characteristics (axial, radial and tangential velocity distributions along the radius) are taken as the mean radial inflow for a steady computation. A set of constrains is obviously set in order to satisfy the required performances (basically, required thrust at a given speed allowing a shift of ±2.5% to speed up the convergence, and blade robustness), while the objectives of the optimization are an increase of efficiency and a reduction of cavitation extent, at both the design conditions.

3. OPTIMIZATION ACTIVITY

3.1 Original Propeller characteristics

The propeller considered for present study is a conventional 4-bladed CPP for a twin screw ship, whose main characteristics are reported in following table 1, where D is propeller diameter, P0.7 is pitch at 70% radial position, d_{hub} is hub diameter, A_E and A_0 are propeller expanded area and disc area, Z is the number of blades.

Propeller characteristics	
D [m]	4.60
$P_{0.7}/D$	1.08
d_{hub}/D	0.30
A_E/A_O	0.72
Z	4

Table 1 - Propeller characteristics

As anticipated, the propeller is operating at constant revolution rate (about 180 RPM) in correspondence to very different ship speeds (24 and 11 kn) by means of blade pitch angle variation (indicated as "reference pitch" and "reduced pitch" respectively in the following).

3.2 Optimization

For the design of the new propeller, different optimization approaches of increased complexity have been applied, including at each step new propeller parameters. At the end, the parameters investigated were:

- global parameters considered in usual propeller design (chord, maximum thickness, maximum camber and pitch distribution along the radius)
- sectional parameters, i.e. camber and thickness distribution along the chord

The number of free parameters included into the optimization process varies between 21 to 30. In all the cases, an initial population of 300 members was considered and let evolved for 100 generations, for a total number of about 100000 different geometries.
Figure 3, for instance, shows a typical plot of the characteristics, in terms of cavity extension, of all the designed propellers obtained during the optimization having, as free variables, the global parameters plus the profile mean line. The main objectives of the optimization, back cavity area at the design pitch and face cavity area at the reduced pitch respectively, are reported on x and y axis, while bubble radius and colour monitor back cavity area at the reduced pitch (not evident for the original propeller and thus to be avoided) and face cavity area at the design pitch (not evident for the original propeller and thus to be avoided).

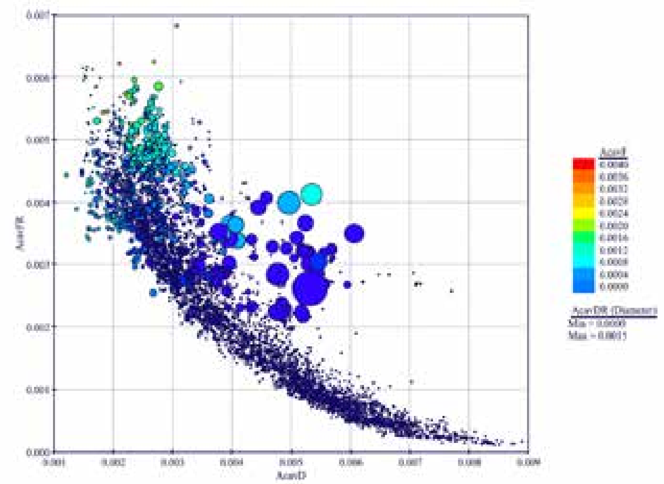


Fig. 3 - Pareto designs for the global parameters plus mean line optimization [Bertetta et al 2012]

In the light of the results of the different optimization approaches, the optimal geometry has been selected among the pareto designs that satisfy the thrust constraints and grant zero face cavitation at the design condition and zero back cavitation at the reduced pitch condition. With respect to the original geometry, all the Pareto designs allow to sensibly reduce face cavitation, only with a minor reduction of back cavitation, consistently with the fact that the original propeller design was centered on maximum speed condition. The new propeller has been selected, among the Pareto solutions, as a compromise between back and face cavitation, having in mind also the side effects of cavitation (in terms of radiated noise). In particular, as mentioned, face cavitation at reduced pitch was definitely the most trying phenomenon for the original propeller, thus it was accepted to have a certain increase for back cavitation at design pitch, obtaining a large reduction of face cavitation. The original and optimized propellers are numerically equivalent in terms of working points. Thrust curves, from which ship speed, at a fixed propeller rate of revolution, depends, are overlapped within the complete range of advance coefficient, as shown for example for the reference pitch in Figure 4. In the same figure, a slight increase in the efficiency is also visible. With respect to the original propeller, face cavitation of the selected optimum propeller was numerically reduced of about 50% (Figure 6) while back cavitation was the 35% greater than the original one (Figure 5).

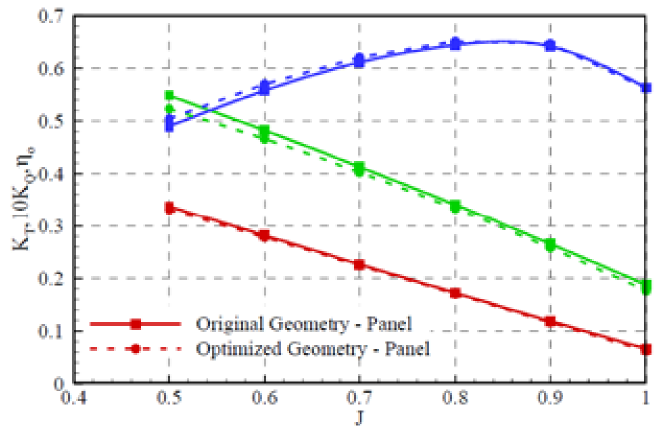


Fig. 4 - Comparison between original and optimized propeller – numerical open water tests – reference pitch [Bertetta et al 2012]

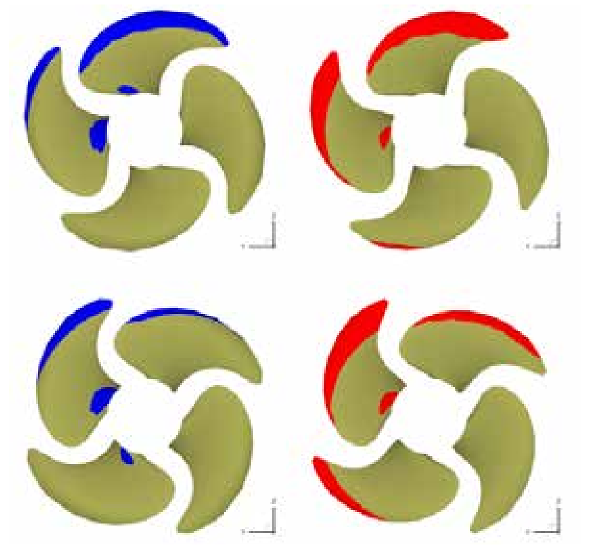


Fig. 5 - Comparison of predicted unsteady back cavity extension (0° - 60°) between original (left, blue) and optimized (right, red) propeller at reference pitch [Bertetta et al 2012]

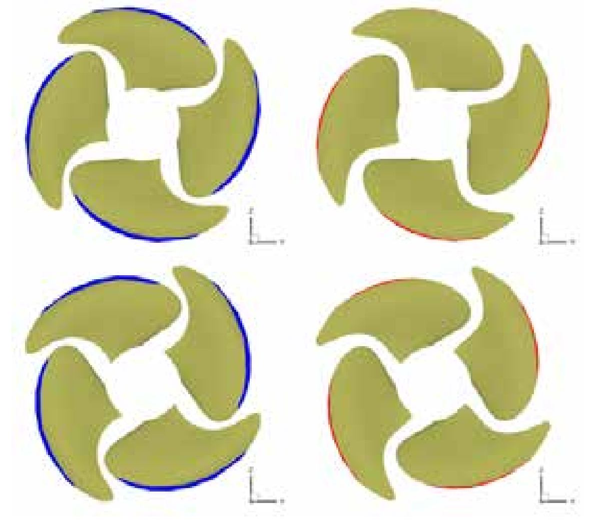


Fig. 6 - Comparison of predicted unsteady face cavity extension (0° - 60°) between original (left, blue) and optimized (right, red) propeller at reduced pitch [Bertetta et al 2012]

Case Histories

4. EXPERIMENTAL CAMPAIGN

4.1 Open Water tests

As a first step in the experimental campaign, propeller open water tests have been carried out at CEHIPAR towing tank for both propellers and pitches, in order to verify their hydrodynamic characteristics. The results, reported in Figures 7 and 8, confirm that the two propellers are equivalent in terms of functioning point at reference pitch (same thrust at same advance coefficient, thus leading to same speed at same RPM), with only a small change (completely acceptable) at reduced pitch. Moreover, in both cases propeller efficiency is increased with the optimized geometry, confirming the numerical results.

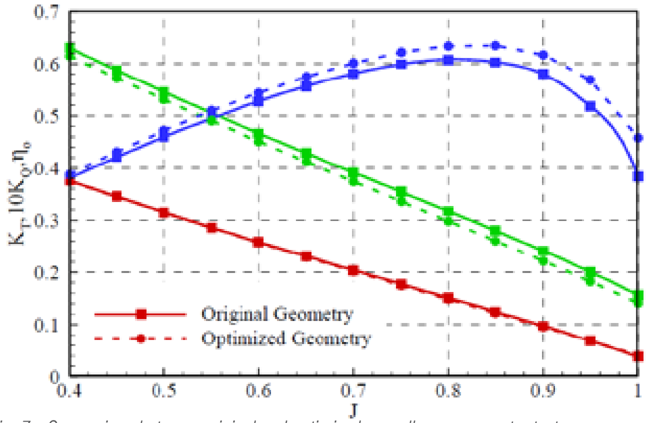


Fig. 7 - Comparison between original and optimized propeller – open water tests – reference pitch [Bertetta, D., Savio, L. and Viviani, M. 2011. Experimental characterization of two CP propellers at different pitch settings, considering cavitating behaviour and related noise phenomena, SMP 2011]

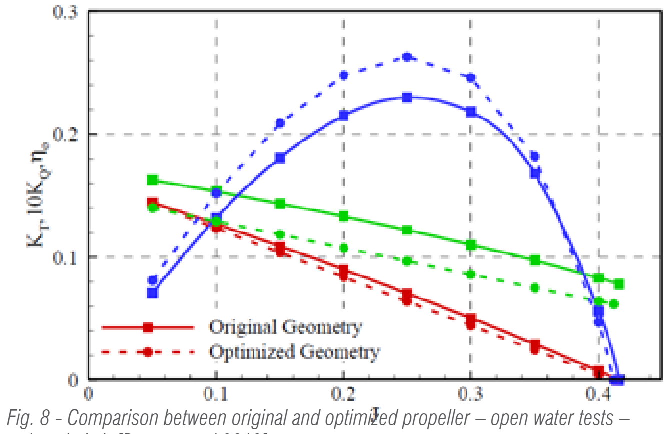


Fig. 8 - Comparison between original and optimized propeller – open water tests – reduced pitch [Bertetta et al 2012]

4.2 Cavitation tests

Cavitation tests were performed at DITEN Cavitation Tunnel, where inception points of various cavitation phenomena have been measured, and cavitation extent observations have been carried out in correspondence to different functioning points. Tests were carried out in correspondence to the nominal wake considered in the design activity and also to a configuration with inclined shaft only, in order to reduce background noise in the tunnel for noise measurements.

In the present work, only the results of the cavitation observations in correspondence to the nominal configuration are reported for the sake of brevity, while complete results are reported in (Bertetta et al 2012).

Results from the optimization activity are confirmed, with slightly worse performances in correspondence to the reference pitch and better performances in correspondence to the reduced pitch. In particular, at reference pitch (Figure 9), a larger chordwise extension of cavitation at higher radiuses towards the tip for the optimized propeller is present; contemporarily, radial extension of cavitation itself is slightly lowered for the optimized propeller. The higher propeller loading results also in the presence of sheet cavitation in correspondence to a wider range of blade angular positions, also outside decelerated wake, confirming the numerical results.

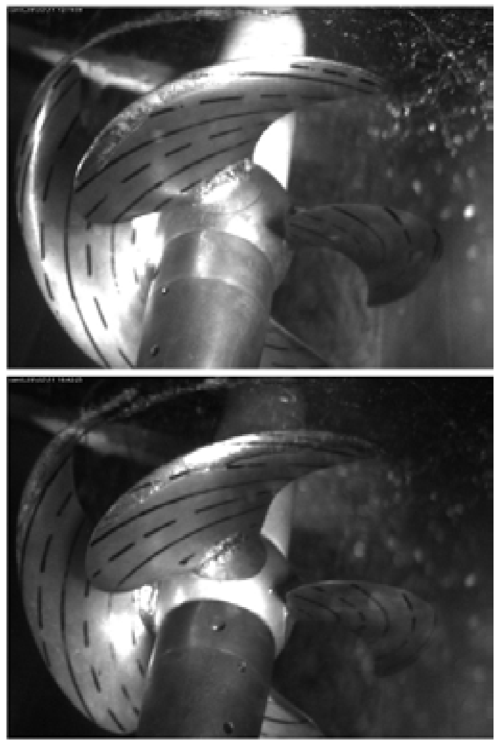


Fig. 9 - Observed cavitation extent at reference pitch Original propeller (above) vs optimized propeller (below) [Bertetta et al 2012]

As expected, the most significant differences are encountered at reduced pitch (Figure 10), as predicted numerically, with a considerable reduction of face related phenomena.

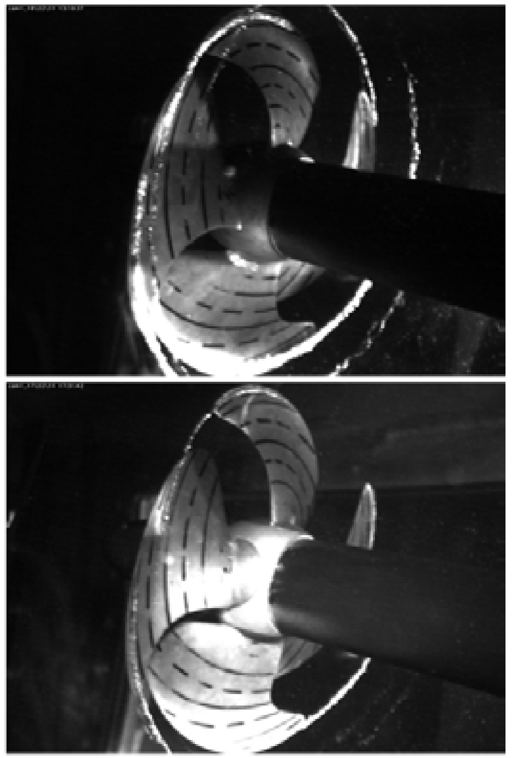


Fig. 10 - Observed cavitation extent at reduced pitch Original propeller (above) vs optimized propeller (below) [Bertetta et al 2012]

4.3 Radiated noise measurements

Radiated noise measurements were carried out in correspondence to a rather large amount of functioning points for both pitch settings and testing conditions (Bertetta et al 2011). The results for the design points only are reported, since the main aim is the validation of the optimization process described above.

The measurements were carried out by means of a Reson hydrophone TC4013, coupled with a Bruel and Kjaer 2635 charge amplifier. The hydrophone has been located inside the cavitation tunnel, outside the direct propeller slipstream. Since water quality is of great importance for cavitation tests and, as a consequence, for noise measurements, during all tests, oxygen content was continuously monitored, as suggested by ITTC, by means of an ABB dissolved oxygen sensor model 8012/170, coupled with ABB AX400 analyser. Constant testing conditions (i.e. oxygen content equal to 40% of the saturation value at atmospheric pressure) were utilised in order to have a fair comparison of the two propellers.

The results are presented in 1/3 octave form. In particular, for each band, the non-dimensional value Kp is evaluated as follows, together with the corresponding level, following:

$$K_p = \frac{p_{rms}}{\rho n^2 D^2} \qquad L_p(K_p) = 20\log_{10}\left(\frac{K_p}{10^{-6}}\right)$$

in which p_{rms} is the root mean square value of each spectrum component. Measurement have also been scaled at a reference distance of 1 m, using the formulation in accordance to ITTC (1978). Finally, in correspondence to each functioning point the background noise is evaluated by repeating the measurements with all equipment running and with the propeller substituted by a dummy model. Net sound pressure levels may, then, be evaluated as suggested in ITTC procedures. If the difference between the propeller noise and the background noise is lower than 3 dB, no curve is represented in the graphs. Figures 11 and 12 present radiated noise measurements for the two propellers in correspondence, respectively, to the reference and the reduced pitch. As visible, the measurements at reference pitch, due to the background noise, allow to characterize propeller noise only below 1 kHz. Nevertheless, this is the most significant frequency range, where the effect of the cavitating tip vortex is predominant. In particular, it is clear that, consistently with the design assumptions, tip vortex related phenomena are amplified in the case of the optimized propeller, showing an increment of about 4 dB of the peak value. Main differences are due to the vortex related phenomena also in correspondence to reduced pitch condition (in this case face vortex and vortex from sheet face phenomena are present), with opposite effect with respect to design reference pitch condition. In particular, it is clear that the delay of these phenomena results in a considerable reduction (about 9 dB) of the noise spectrum of optimized propeller with respect to the original one. All these results are clearly in line with design goals, with a voluntary slight worsening of propeller behavior in correspondence to the reference pitch in order to reduce significantly noise at the reduced pitch, which was considered as the most important problem for this propeller. This therefore confirms the validity of the proposed design approach.

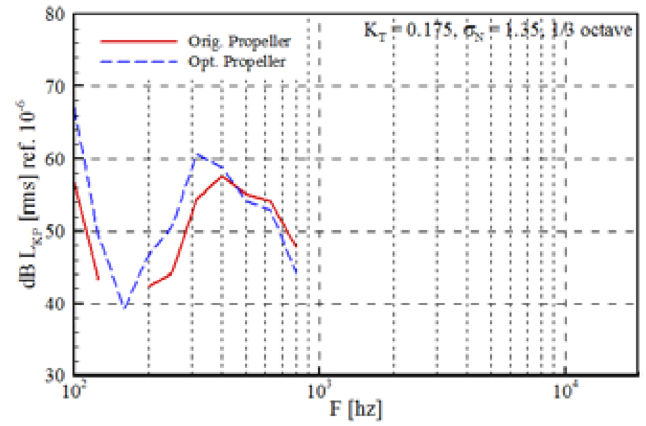


Fig. 11 - Radiated noise measurements at reference pitch [Bertetta et al 2012]

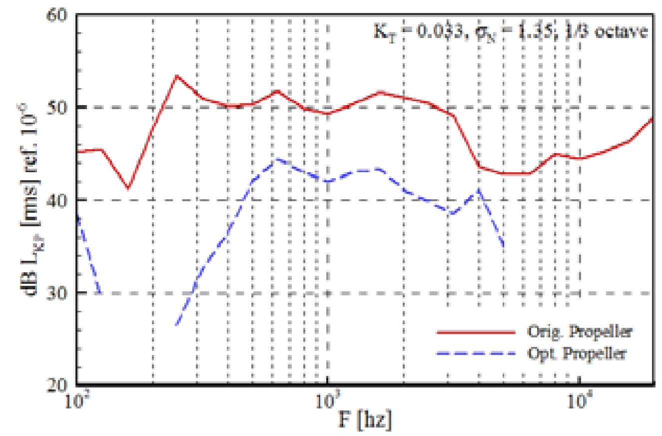


Fig. 12 - Radiated noise measurements at reduced pitch [Bertetta et al 2012]

5 CONCLUSIONS

The design of a CPP propeller has been carried out, utilizing a cavitating panel code for propeller analysis coupled to modeFRONTIER. A challenging task has been considered trying to optimize propeller behavior in correspondence to two very different conditions characterized by reference and reduced pitch.

The goals of the design (improvement at reduced pitch without deteriorating too much the reference condition) were satisfactorily achieved; the successive experimental campaign, moreover, confirmed the predicted propeller behavior. The optimization tool proved to be an efficient mean to improve propeller characteristics, even if in a such problematic case it is difficult to obtain very large improvement at both operating conditions with controllable pitch propellers if operated at constant RPM. In case the same tool is applied to a single operating point, it is expected that much larger improvements may be obtained.

ACKNOWLEDGEMENTS

Part of the activities presented in this work have been carried and funded by SILENV - European Collaborative Project n° 234182.

S. Gaggero & M. Viviani
Department of Electrical Electronic, Telecommunication Engineering and Naval Architecture (DITEN), Genoa University, Genoa, Italy
D. Bertetta
Naval Architecture Department, Naval Vessel Business Unit, FINCANTIERI S.p.A., Genoa, Italy, formerly Genoa University
S. Brizzolara
MIT-iShip, Innovative Ship Design Lab, Department of Mechanical Engineering, Massachusetts Institute of Technology, formerly Genoa University
L. Savio
Ship and Ocean Laboratory, MARINTEK (Norsk Marinteknisk Forskningsinstitutt AS), Trondheim, Norway, formerly Genoa University

Case Histories

Comparative analysis of temperature control systems for **high pressure die casting dies**

In High Pressure Die Casting production it is necessary to keep the die temperature within a certain specified range for the following basic reasons:

- Ensure that the casting solidifies progressively from the opposite part to the running systems, in order to reduce much more as possible the porosity.
- Reduce hot spots and make the die more isothermal.
- Modify the grain size and microstructure to improve mechanical properties.
- Increasing the die life time.

Working with a die at excessively low temperature there will be problems like premature solidification and incomplete filling of the die cavity, difficult ejection due to increase of shrink force and in consequence of high thermal gradients, rapid die wear, thermal shock of the die surface and also formation of cold shuts.
If the die temperature is too high there will be a fast release agent degradation and increased consumption, longer cycle time, unreliable casting dimensions and an increase of shrinkage porosities. Furthermore the soldering defect can appear, thereby resulting in low quality of the component and difficult ejection.
As a result, the correct die temperature is crucial to obtain a smooth and high level of productivity and to optimize the production cycle.
Numerical methods for the thermal analysis of the HPDC process are needed to shorten the cost, the design and production time and the samples to involve in finding acceptable operating conditions. CAE tools, like MAGMASOFT and its Optimization module MAGMAfrontier, allow to support the designer during the research of the best engineering solution.
Temperature Control Units (TCU), or Thermoregulators, are electrical mechanical devices designed to regulate dies temperature.
Thermal regulation takes place by heating or cooling a fluid, which is pushed through an electric-driven pump inside cooling channels within the die. Heat transfer is by convection between the fluid and the surface of the channels in the die and by conduction within the die steel itself.
Standard water TCUs are open tank systems which can control water temperature lower than 90°, characterized by low cost and low temperature range. Pressurized water TCUs are closed tank system which can control water temperature near to 180°C, 200 in some cases. These systems combine the possibilities to preheat the die to high temperature and to cool it efficiently with water.
Other commonly used fluids are diathermic oils, which allow to reach a maximum temperature of 350°C.
With the use of CAE software it is possible to understand the difference between the use of oil and water.
It's important to notice that there are also localized temperature control systems like bubblers, baffles, heat pipes and Jet Cooling systems; these are useful to control specific zones of the die, the "hot spots".
This works aims to select the best temperature control technologies for the most recurring high pressure die casting applications. In order to achieve this purpose, the die casting processes and their temperature control systems have been studied. Furthermore, process simulations with the software MAGMASOFT have been useful for the final comparison.

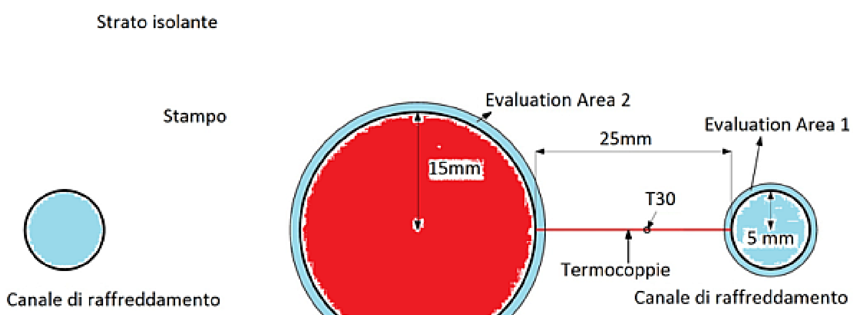


Fig. 1 - Representation of the geometry example

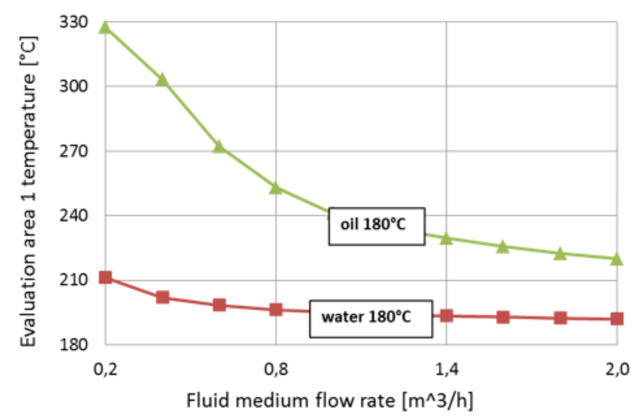


Fig. 2 - Temperature of Evaluation Area 1at 20th cycle versus fluid medium flow rate

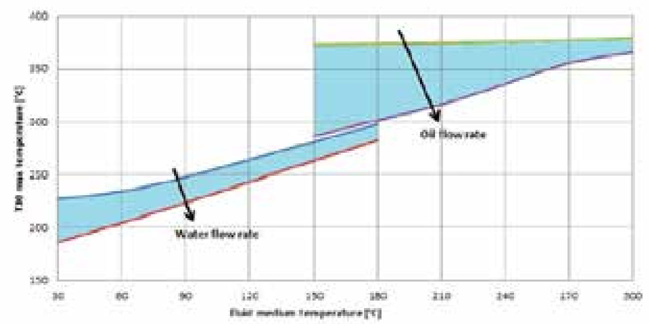


Fig. 3 - Maximum temperature of the thermocouple T30 at 20th cycle versus fluid medium temperature, flow rate from 0.2 m³/h to 2.4 m³/h

Simplified geometry process simulations

The first step of simulations with the software MAGMASOFT has been done on a test geometry to better understand the difference between water and oil used as fluid media. For the thermal analysis of the mold has been considered a steady state simulation.
The influence of the external environment has been denied with an insulation layer around the die and the lubrification time set at zero seconds. Thus, the thermal regime of the die depends only on the presence of the thermal regulation channels (Fig. 1).
The result of this first step is: the die temperature, using water as cooling medium at the minimum flow rate of 0.2 m³/h, is lower than the one using oil at 2 m³/h.
Moreover, the influence of the water flow rate is negligible when the values are higher than 0.8m³/h, while it's still appreciable for oil.
The final result is that water reaches an efficiency saturation point faster than oil for its better thermal properties, as a result, the difference between the two fluids is higher when the flow rate is lower (Fig. 2).

Die temperature vs. fluid medium temperature

The thermocouple T30 was inserted in the die 15mm away from the cavity surface to monitor its temperature.
With flow rates set at maximum, 2.4 m³/h, the maximum temperature at the 20th cycle was recorded. This was repeated for flow rate set at minimum, 0.2 m³/h, for temperature from 30° C to 180° C for water, and temperature from 150° C to 300° C for oil. Fig. 3 shows the effect of fluid medium temperature in steady state temperature of the die, the arrows show the effect of the flow rate.
Oil at 0.2 m³/h has no influence in the die temperature, while water at minimum flow rate cools even faster than oil at maximum flow rate, it's evident in the temperature zone from 150° C to 180° C (Fig. 3.)

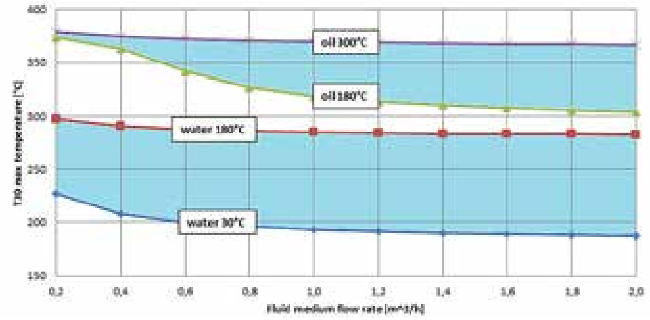


Fig. 4 - Maximum temperature of the thermocouple T30 at 20th cycle versus fluid medium flow rate

Die temperature vs. flow rate

To better understand the effect of the flow rate, at predefined temperature for water, 30° C and 180° C, and oil, 180° C and 300° C, the maximum temperature at the 20th cycle was recorded.
The results (Fig. 4) show that the temperatures of the die decrease rapidly when flow rate increase while its value is still low. However, the decreasing rate slows down with increasing of flow rate and it results negligible for higher values. This is due to the fact that, the flow rate of the fluid medium is directly connected to the heat transfer coefficient for convection, but it rules only in the proximity of the cooling channel.

Real casting process simulations

During a real die-casting process there are many kind of boundary conditions, such as die/casting, die/air, die/cooling channel and die/lubrication. In these simulations they are considered and treated by heat-transfer coefficients across the specific interfaces.
The cycle parameters, shot curve and spray time are constant to investigate the effect of the thermoregulation but the opening time is not the same for every configurations. The difference of solidification time between water

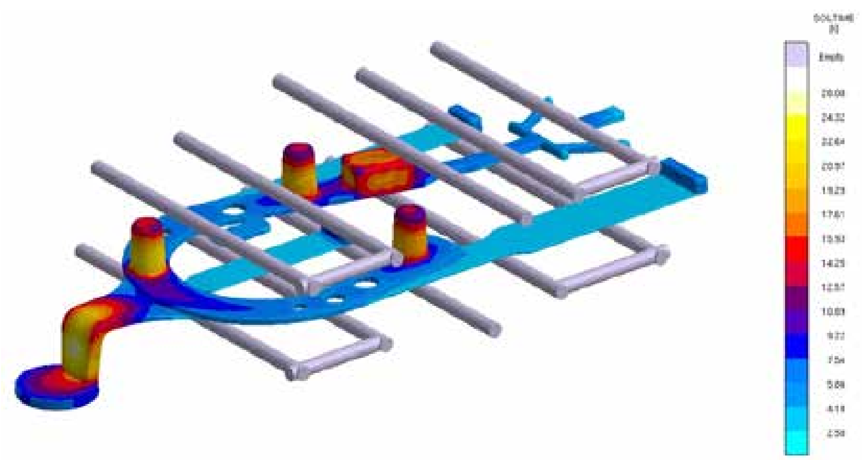


Fig. 5 - Solification time at 15th cycle

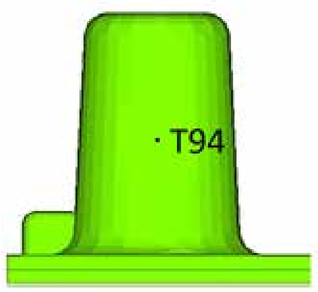


Fig. 6 - Position af the thermocouple T94

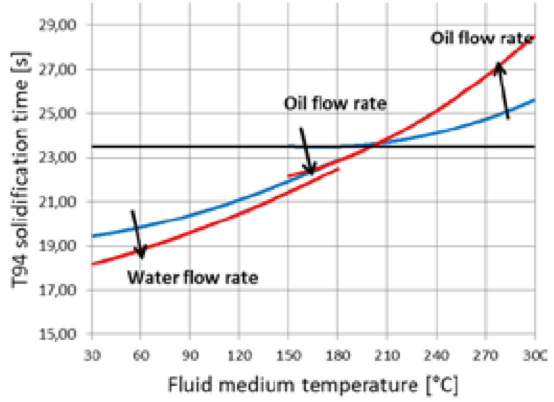


Fig. 7 - Solidification time of the casting versus fluid medium temperature (blue lines mean 0.2 m³/h, red lines mean 2.4 m³/h, black line means no cooling channels)

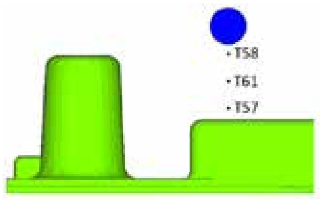


Fig. 8 - Position af the thermocouple T94

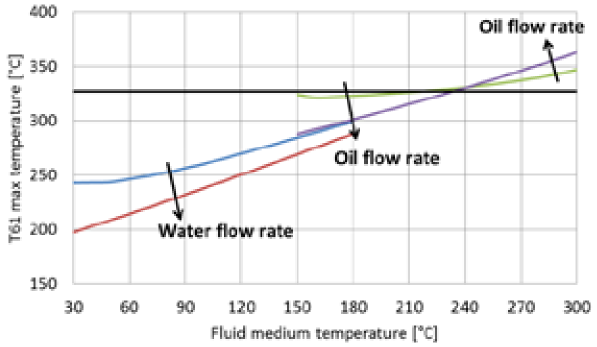


Fig. 9 - Maximum temperature of thermocouple T61 versus fluid medium temperature, flow rates from 0.2 m³/h to 2.4 m³/h

and oil is too high, thus the die opens when the virtual thermocouple T94 in Fig. 6 reach the temperature of 478°C, one degree below the solidus temperature for the aluminium alloy AlSi9Cu3 used.
The geometry used for these simulations, illustrated in Fig. 5, is part of the European Project Music and it has been shaped to create defects and to learn how to reduce them.
It's possible to see in Fig. 5 that the difference in solidification time between the thick zones and the thin zones around is solidified to 10 seconds. This means a high probability of shrinkage porosity defects inevitable with the only use of thermoregulations. Possible solutions to the problem could be the introduction of localized temperature control systems, like jet cooling, or the adjustment of the casting geometry, or the introduction of the local squeeze to compact the alloy.
The virtual analysis highlighted that the use of water can reduce the cycle time for more than 4 seconds (Fig. 7). On the other hand, with fluid medium temperatures higher than 200°C the cycle time is longer compared to the condition without cooling channels (black line), in fact these temperatures are usually used only for preheating.
Flow rate variation brings a maximum solidification time variation of one second for water. Thus, the temperature of the fluid is much more important than its flow rate; it brings a solidification time variation of 4-5 seconds for water and 7-8 seconds for oil.
It's, also, evident that an increasing flow rate brings to lower die temperature except when using oil at temperatures higher than 230°C. When the temperature range is from 150°C to 180°C both water and oil can be used, the difference is a much higher heat transfer coefficient for convection of water, translatable as lower temperatures of the die. Fig. 9 highlights the possibility to predict the die temperature on varying of the flow rate and the fluid medium temperature. Thus, it's possible to choose the operating parameters to obtain the desired temperature profile within the die. A preheated die involve longer die life. Moreover, Fig. 10 shows that the steady state temperature condition is achieved faster with a preheated die; it means a potential reduction in rejected casting of 50%, even better with higher preheating temperature.

Conclusions

The ability to define the correct operating temperature of the molds can provide greater assurance of dies life while checking the quality of the castings produced.
The thermal control of the High Pressure Die Casting process is very complex and run in a single cycle time by a multitude of variables.
The work done allow to identify not only the dynamics of the cooling function of the systems used (water or oil) but also to try to give some indications of use as indicated below.
Some subjective judgments have been necessary to solve a very complex problem with a simple matrix. Actually, CAE tools like MAGMASOFT and its optimization module MAGMAfrontier, are needed to select the solution for each particular application. Nevertheless, some guidelines could reveal useful if those instruments are not available, or in order to direct the analysis on a more restricted number of alternatives.

Paolo Callegaro, University Padova
Giampietro Scarpa, EnginSoft

For more information:
Giampiero Scarpa, EnginSoft
g.scarpa@enginsoft.it

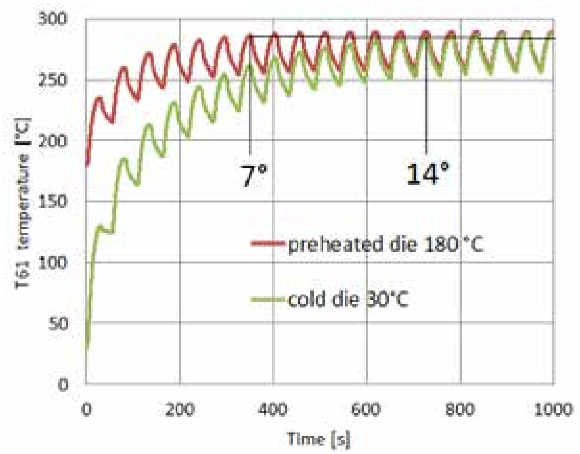


Fig. 10 – Temperature distribution during the dies heating

Development of an experimental/numerical methodology to evaluate the **comfort level of high-heeled shoes**

The WIN-Shoes project, conceived in the Tuscany shoe district, aims to tackle a crucial problem faced by SMEs working in this field, where new expectations of the globalized market rule: "innovation" is required to lead in the market and be competitive without reducing the level of quality and fashion expected by Tuscany shoes producers.
WIN-shoes was developed after the success of the related ECO HT-Shoes project in product innovation. This led to the creation of the new brand "Made in Tuscany" and a totally new trademark: "Violavinca". After such success the WIN-Shoes partners decided to continue their innovation process with a radical change in their whole manufacturing and organizational operations.
The idea of the WIN-Shoes project was to radically change the way SMEs work in this sector, by moving them from a traditional approach to a more technological one. This was done by adopting technologies normally used in other industries to positively impact shoe production. The project considered the development of a design ICT platform that uses CAS/CAD/CAE/CAM tools in order to digitalize both the design process and the prototyping phase. The platform consists of "functional modules" devoted to specific tasks, and includes a new working method that can be defined as a "new process for shoe development". This will allow companies using the platform to achieve a clear improvement in competitiveness, reduced cost and time when designing and manufacturing their new collections. This is all done while increasing the products quality, that will be optimized in relation to technological, structural, comfort and price parameters. Figure 1 shows the workflow proposed by the project.
WIN-Shoes stands for "When the INnovation makes Shoes" and it is a research project co-funded by the Tuscany region and the European Union. The involved partners are: two shoes producers, Calzaturificio Everyn and Calzaturificio Maruska, four research institutions in the framework of Archa Lab, BioRobotics Institute Scuola Superiore Sant'Anna, University of Pisa and EnginSoft, and finally Tuscany Service and Rete Leonardo for company networking – Figure 2.

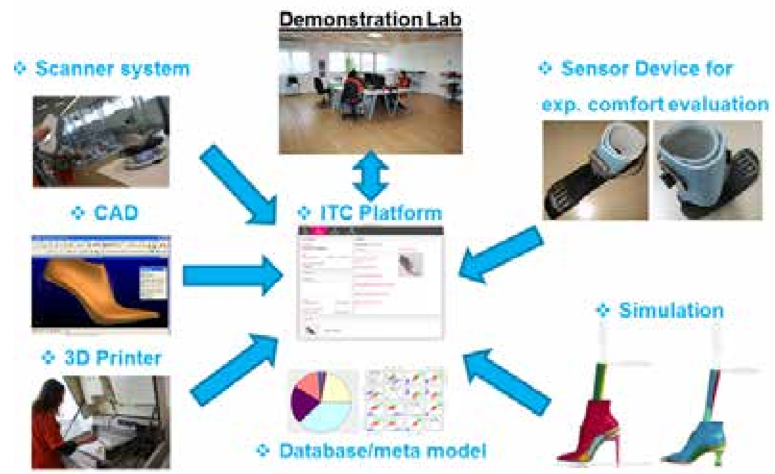


Fig.1 – Final workflow of WIN-Shoes project

Fig. 2 – WIN-Shoes partners

Methodology

Numerically speaking, the simulation of a person wearing a shoe while standing is not a simple problem to solve. One of the main difficulties is related to the precise position of the foot inside the shoe, which is determined largely by the brain that controls the muscles and tendons stiffness and by the preload that influences the foot and the articulation orientation. This is all done automatically by the body to allow the shoe to be worn in the most comfortable way. Another complex part is the setting to be applied to the single parameters, which cannot be automated due to the time-consuming procedure necessary for each simulation.

Due to the complexity of this analysis, it has been extremely important to create a suitable workflow to manage the data and to elaborate a series of numerical "artifices", allowing an accurate simulation of the scenario. The methodology finally proposed and validated has been defined after a long exploration process of several software and numerical strategies.

EnginSoft has dealt with the numerical part of the project, by developing a simulation methodology that uses the explicit finite element code LS-DYNA to represent shoe wearing and shoe loading in the standing condition. Starting from CAD (provided by shoes producers) and experimental data (obtained by static tests performed by the BioRobotics Institute SSSA) the methodology has been applied and two biomechanical models have been generated (constituted by bone structure, soft tissue and heeled shoes). In order to model the foot wearing on the shoe, the interpenetration between the upper shoe and the foot had to be removed, therefore a kinematic approach has been used, with the purpose of enlarge, mechanically, the upper shoe itself. The motivation for using LS-DYNA was to avoid any convergence problem with contacts and large displacements . The synthesis of the workflow for the proposed methodology is shown in Figure. 3.

The workflow can be summarized as follows:

- Feet scanner and feet bones CAD generation -> Bones CAD and feet CAD gathering -> Simplified bones CAD generation -> Simplified bones alignment and generation of a new simplified feet/bones CAD -> Simplified feet/bones FEM model generation.
- CAD Shoe creation -> FEM shoe generation with enlargers -> Application of the enlargers movement.
- FEM models gathering.
- Simulation/Tuning.

Simulation

Once the FEM models of the shoe and the foot are generated, they are assembled and the boundary conditions are applied, ready for the analysis to be performed. The foot is already assumed to be in the right position as inside the shoe.

The simulation of the shoe fitting and of the shoe static load has been divided into 5 phases:

Phase 1 – Upper shoe enlargement. The elastic enlargement of the upper shoe is performed differentiating the movement of the virtual tools, called enlargers, till any interpenetration between foot and upper shoe is eliminated. During the upper shoe stretching, only few contacts are actives, these are: enlargers/upper shoe, internal sole/foot and finally bones/foot. Only the last ones are active during the whole simulation.

Phase 2 - Upper shoe release. In this phase the contact between foot and upper shoe is activated, while the enlargers are closed again so to allow the upper shoe to lay down on the foot.

Phase 3 – Oscillations stabilization. No activity is performed. The model is allowed to settle, till no vibrations related to the analysis process are present.

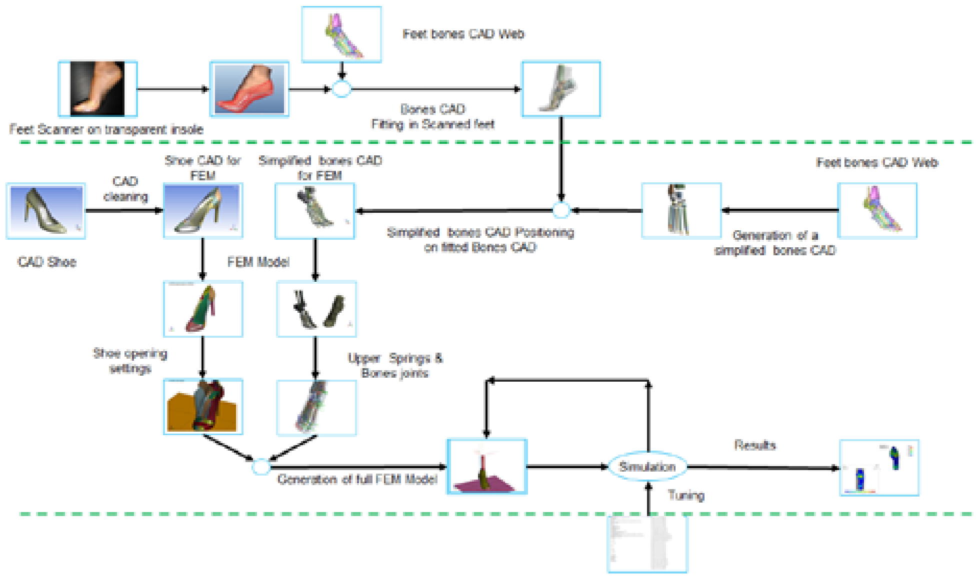


Fig. 3 – Workflow Methodology

Phase 4 – The constraint on the upper part of the sole is eliminated, while the contact between the ground and the external sole is activated.

Phase 5 – Static load. A movement is applied to the bones, so to move the foot downwards, till the weight rate loaded on the foot is reached.

The comparison between the simulation and the experimental test is performed by comparing the pressure distribution elaborated during the static test on the sole and between fingers and upper shoe, using equivalent magnitudes as calculated with the virtual bio-mechanical model.

The calibration procedure consists of an iterative non-automatic process where some physical and numerical parameters are manually modified in order to make the simulation control magnitudes comparable with those used in the experimental tests.

Fig. 4 – Virtual bio-mechanical models mod.1-5

Results

Five virtual bio-mechanical models related to five different shoe models have been elaborated, each of them corresponding to a full model as in Figure. 4. The models considered as the most critical ones to be worn are nr. 1 and 2 (information obtained by experimental tests on comfort). These models have been analyzed using the proposed methodology and the following figure show the comparison between experimental data and numerical results, for model nr. 2, considering the plantar pressure distribution (Figure. 5) and the contact pressure between fingers and upper part of the upper shoe (Figure. 6). The results indicate a good alignment between experimental data and simulation values, proving that this methodological approach, applied to both the simulation procedure and the calibration, is suitable for industrial applications.

Conclusions

The activity performed in the numerical context has developed a methodological approach to virtually reproduce the static test of shoe fitting. Five virtual biomechanical models of the foot have been created, in relation to the corresponding shoes and the two most critical ones have been analyzed applying and validating the proposed methodology. In the end, as far as the numerical part is concerned, further developments could be foreseen:

- Results tuning, working out a new biomechanical models with separates fingers and including the ligaments which connect the bone structure of the foot.
- Evaluating a methodology to simulate the walking action.
- Improve the whole workflow. The current procedure requires the constant work of the analyst, therefore it could be interesting to investigate the possibility of modifying the methodological approach in order to automate the whole workflow, Starting from the foot in relaxed position and modeled completely with muscles/tendons according to literature, the possibility of moving the joints of the bones in an automatic/iterative way, with the purpose of positioning the foot in any shoe models.

	Experimental (KPa)	Simulation Mean (KPa)
S1	30	23
S2	10	14
S3	10	13
S4	35	43
S5	140	145

Fig 6 - Comparison between the experimental/numerical data between fingers/upper shoe pressure for shoe nr. 2

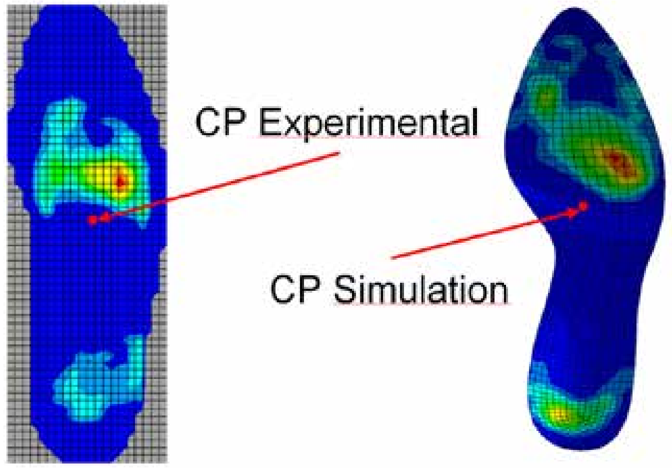


Fig 5 – Comparison between experimental plantar pressure (left) and numerical plantar pressure (right) for shoe nr. 2 and CP – Centre of Pressure

- Use the gained experience in this specific sector for any other possible application in the fashion sector, following the same philosophy. It might be interesting, for example, using virtual mannequins of different sizes, to simulate garments wearability for different physiques or analyze how a fabric, under the action of intelligent fibres, changes its shape so to better suit the body size.

Massimo Tomasi, Valentina Peselli - EnginSoft
Francesca Gambineri Archa
Stefano Roccella The BioRobotics Institute SSSA

For more information:
Massimo Tomasi, EnginSoft - m.tomasi@enginsoft.it
WIN-Shoes web site : http://www.win-shoes.net

CAE tools application within the SPIA
Innovative Aeronautic Primary Structures

SPIA is an Italian research project, co-funded by MIUR (Italian Ministry of University and Research), which sees some of the most important DTA's (Apulian Aerospace Technological District) Companies working together, with the aim to develop and produce advanced technologies of design and production for composite empennage components (fixed and mobile) and for the relative fuselage segments on which they are installed, in order to:

- improve the quality and reliability of the finished product;
- reduce significantly the time and cost of manufacturing;
- reduce the environmental impact of aeronautical components production.

These activities are focused on innovative technologies regarding sizing, designing and production of components for regional jets.

The EnginSoft activities consisted in the realization of an analytic tool for the preliminary design and dimensioning of fuselage sections. The main aims of the tool are:

- provide a useful set of tools which allow the designer to obtain a configuration trade on fuselage primary structure in the preliminary design activity;
- procure size and margin of safety for every component.

Irrespective of shape, the basic fuselage structure is essentially a single cell thin-walled tube including skin, transverse frames, stringers, and "closed" frames known as bulkheads.
There is a principal method to schematize and analyze a fuselage section: the lumped parameter method.

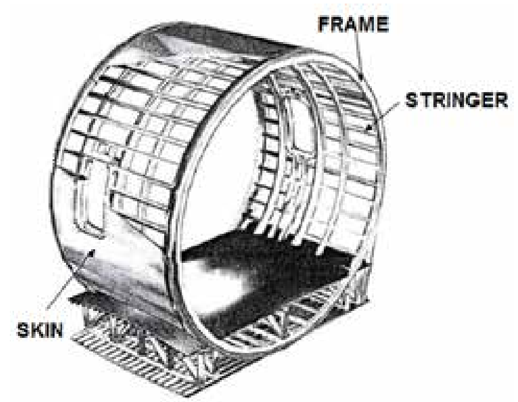


Figure 1 - Typical fuselage section

LUMPED PARAMETER METHOD

An aeronautical beam can be simplified with the concept of thin-walled section. This idealization has three main hypothesis:

- Panels are thin, so they carry only torsion and shear.
- Stringers section is very small compared with the entire fuselage section. They contribute at the global inertial properties of section only in terms of transposed moment of inertia; they carry only bending and axial action.
- Frames keep unchanged the shape of the section in the perpendicular plane. They are taken into account considering only their relative spacing (and so the length of panels).

MODIFIED LUMPED PARAMETER METHOD

The analytic tool developed during this activity is based on this theory, with some differences that improve the obtained solutions:

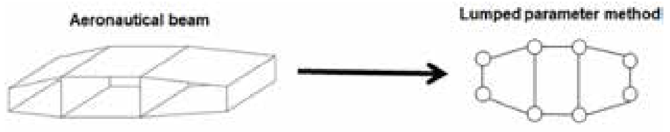


Figure 2 - From aeronautical beam to lumped parameter method

- Panels also endure a portion of the bending moment and the axial load, while they endure shear flows only in the lumped parameter representation;
- Stringers retain their characteristic cross-section, so their second moment of area Ixx;
- the frames are considered not only in terms of spacing, but also taking into account shape, dimensions, thickness and loads absorption.

Aero-DeSTo
(Aeronautical Design Structural Tool)

The tool operates in two different way:

- single launch mode
- optimization mode

In both the cases, the tool provides first the design of skin and stringer, then, starting from the chosen solution, proceeds with the sizing of the frame.
The tool supports the designer during the dimensioning of fuselage section not only in terms of resistance criteria but also in terms of buckling and post-buckling behavior of the components. It is completely customizable as concerns material and it is flexible for any other preliminary design of aeronautical components, such as wing and tail empennage. The fuselage is divided in four different zones: keel under the floor line, crown symmetric respect to the keel, and two sides that are the lateral parts. For any sector the tool verifies the geometric parameters and calculates the margin of safety.

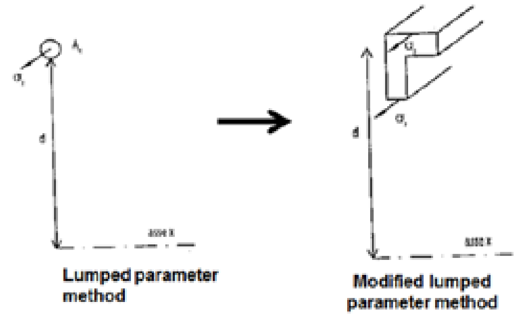


Figure 3 - Modified lumped parameter method

INPUT

Materials
The user has to define the material properties. For every material type (composite or metal) different parameters have to be produced.
Loads
The loads can be applied in terms of shear, normal action and moment about any axis, in the center of gravity of the section. It is also possible to apply concentrate loads in any point of the frames.
Number and dimensions of elements
The other input data, necessary to start the analysis, are:

- the number of stringer in the sectors of the section, (crown, keel e side),
- the thickness of panel (that are indipendent in the 4 noted zones),
- the type and dimension of stringer and frame section.

These parameters are given from the user or directly from the optimization tool.

OUTPUT

Mass: one of the output of the tool is the total mass of the section and the mass of all the sized components.
Margin of safety: once known the stresses on the various components and the stress allowables, it is possible to calculate the margin of safety.
For the sizing of panel the tool calculates the Margin of Safety considering:

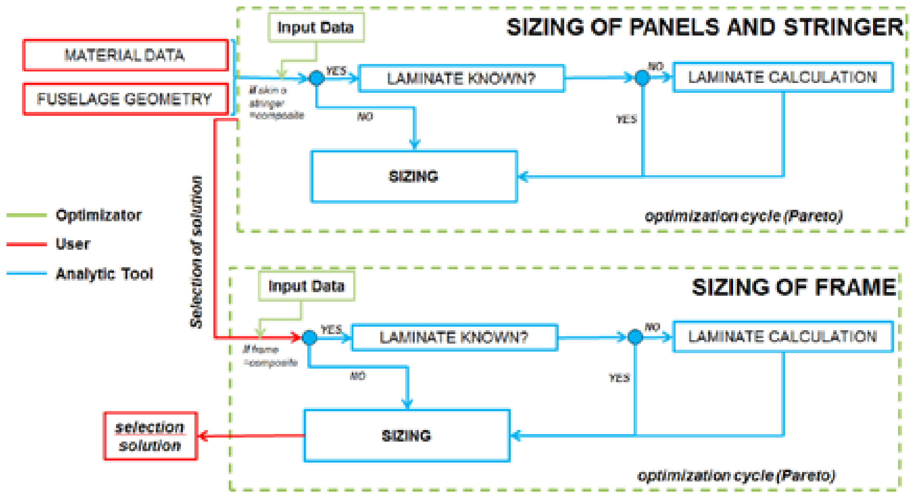


Figure 4 - Tool logic flow chart

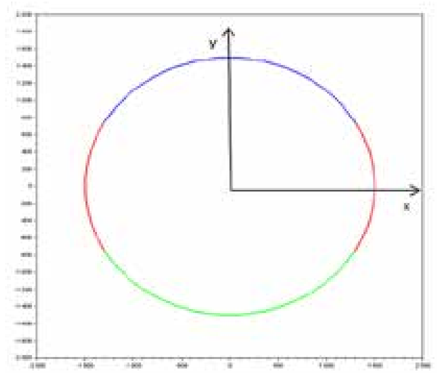


Figure 5 - Fuselage section

- Strenght Criteria (Von Mises, Tsai Wu)
- Buckling and post-buckling criteria (Diagonal Tension, Permanent Buckling).

For the sizing of stringer and frames the tool calculates the Margin of Safety considering:
- Strength Criteria (for beam in traction)
- Crippling
- Column buckling
- Inter-Rivet buckling
- Lateral-Torsional buckling

$$MS = \frac{\sigma_{allow}}{\sigma_{applied}} - 1$$

The Aero-DeSTo tool has been coupled with modeFRONTIER for a preliminary design and optimization of a fuselage section. Once built up the workflow, a first sensitivity analysis has been performed to point out the independent performances and then the multi-objective optimization addressed vs. the 3 more effective objective functions.

All the solutions can be sketched into a 4D Bubble Chart, in which the more conflicting performances lie on x, y axis.

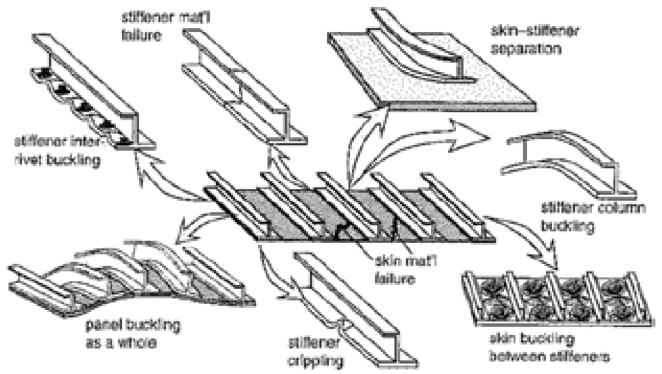


Figure 6 - Failure mode of aeronautical panel and stringer

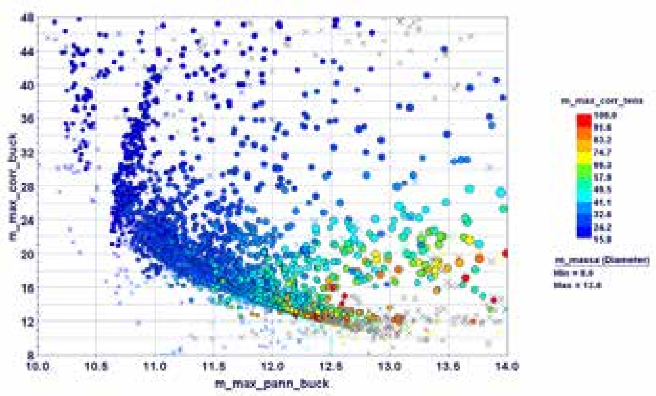


Figure 8 - modeFrontier solution

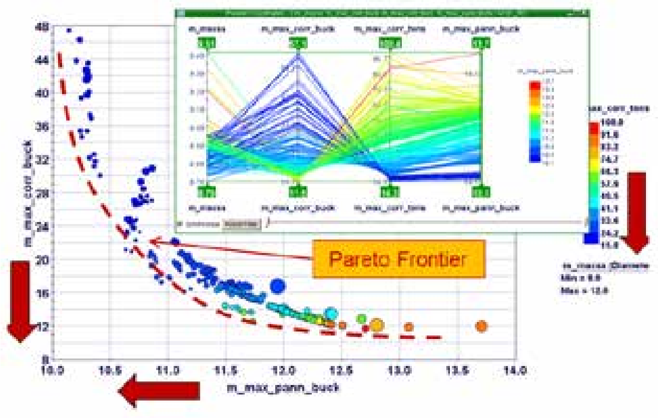


Figure 9 - modeFrontier PARETO solution

The best designs are given by the Pareto Frontier (performances trade-off).
The selection of the candidate designs can be done by suitable filters on Axis Parallel Chart.

CONCLUSIONS
- The tool takes place between the preliminary design phase and a detailed analysis , in the way to overcome the gap between CAD and FEM tools, that today is faced by a trial and error analysis, redesigning all the model in case of a problem of sizing.
- It is suitable to test different configuration in a preliminary design contest, and so to give the possibility to operate a trade-off analysis in this early phase, and to avoid wrong solutions without operate CAD and FEM activities.
- The tool is completely customizable in terms of material and is flexible for any other preliminary design of aeronautical components, such as wing and tail empennage.
- The integration into an optimization process and the low computational cost of the tool itself, produce a series of Pareto optimal solutions that is the best way to operate a choice between different designs.

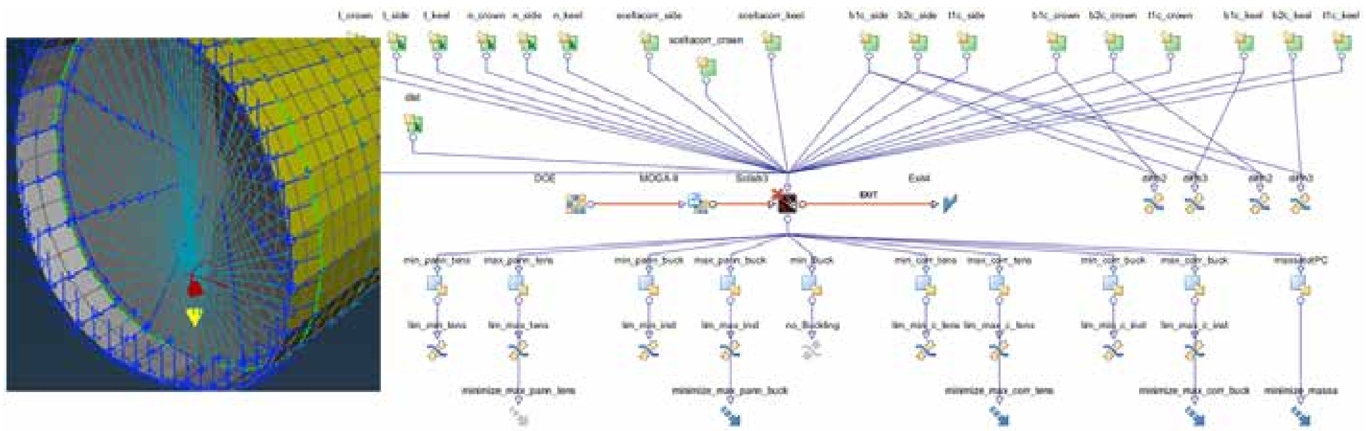


Figure 7 - modeFrontier workflow

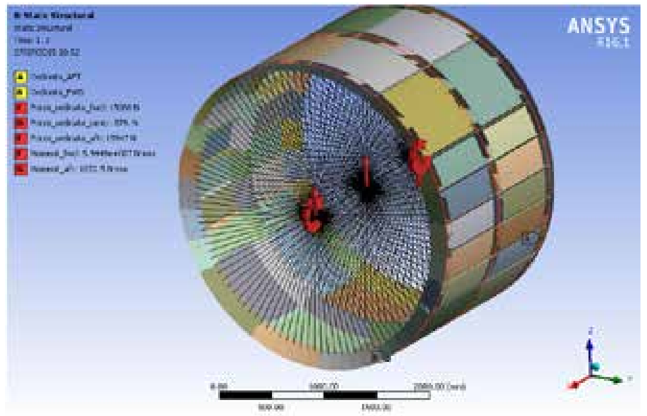


Figure 10 - FEM model of fuselage section

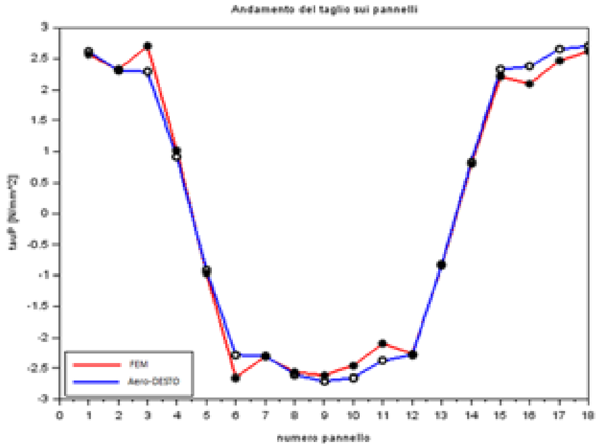


Figure 11 - FEM vs AeroDESTO tool solution

FEM COMPARISON
Aero-DeSTo results have been compared with those coming from a FEM model, for validation purposes.
The FEM model realized represents a fuselage section sized also with the Aero-Desto tool. The skin is made with shell elements, the stringer and the frame with beams.
The comparison results in terms of shear stresses show comparable values between Aero-DeSTo and FEM.

Dario Ricci, EnginSoft
Benedetto Gambino, Finmeccanica

For more information: Marco Spagnolo, EnginSoft
m.spagnolo@enginsoft.it

STAY TUNED ON OUR MUSIC

The MUSIC project (MUlti-layers control&cognitive System to drive metal and plastic production line for Injected Components) is now entering its final stage and it's already working in view of its conclusion. The long research phase has been completed with encouraging results, the experimental tests have provided good feedback and the demonstration activities in the production sites on real production lines are allowing to verify the system functionalities. The consortium is confident that it will achieve the expected objectives and is actively working also in view of their presentation to the next important event chosen for the dissemination of the MUSIC Project, that is at the High Tech Die Casting 2016, that will take place in Venice next June. After the great success obtained in EUROGUSS, in terms of interest and request for information and training on MUSIC topics (such as process control, real-time data monitoring and quality prediction and analysis of casting components, etc.), the HTDC session totally dedicated to MUSIC will be a sort of mini-course delivered by MUSIC partners (EnginSoft, University of Padova, Electronics GmbH, Audi AG, MotulTech Baraldi, University of Aalen) in order to transfer the acquired knowledge, to present real application cases and future developments. This occasion will provide the ideal context to release also the revised and upgraded MUSIC Guide to Key-parameters in High Pressure Die Casting, combining the research results already presented in the first publication, together with the project final results, applications and case studies.
The MUSIC project is also committed in further initiatives oriented to better understand the actual status, features and needs of both the High Pressure Die Casting and the Plastic Injection Moulding sectors on a European level.
Two different surveys are respectively promoted by the consortium through its partners and supporting entities, accessible through the project website: http://music.eucoord.com/Survey/body.pe - http://music.eucoord.com/Plastics/body.pe
Join us and collaborate in these activities! Stay tuned on our MUSIC also on music.eucoord.com

ANSYS MECHANICAL 17.0

ANSYS 17 can be defined as a historic passage, for its greater power, fastness and focus on performances and innovation; it represents a vertical evolution still called "10X" thanks to significant improvements on different topics. 10X as the time reduction for modeling and configuration of FEM models, 10X as reduction of CPU time, management of big models and improvement of multiphysics interaction. It presents new advanced and unique features in FEM environment and, above all, a new concept of simulation and sense of belonging to a community, thanks to the new ANSYS APP Store.

SpaceClaim modeling environment allows to import the geometries produced by other CAD editors, directly in their native format in an even more robust manner. New clean-up and simplification tools have been implemented in addition to those already existent, which facilitate the preparation of CAD for the next phase of analysis in Mechanical environment, drastically reducing the simulation time. In this sense a remarkable step forward has been made in the automatic conversion of solid models with beams and shells, which is a essential feature for the efficient management of big assemblies.

In SpaceClaim some specific tools have been introduced for Reverse Engineering and Additive Manufacturing, which allow to import models in STL format. It is possible to manage the solution files produced in ANSYS (deformed mesh import), meshes developed with other simulation software, from laser acquisition, etc. Beyond the conversion of STL files into 3D geometries, it is possible to do the opposite operation, generating a STL file readable for 3D printing machine. By the way, specific tools permit to create complex geometries for Additive Manufacturing process, considering organic-shape fillings or core with regular patterns.

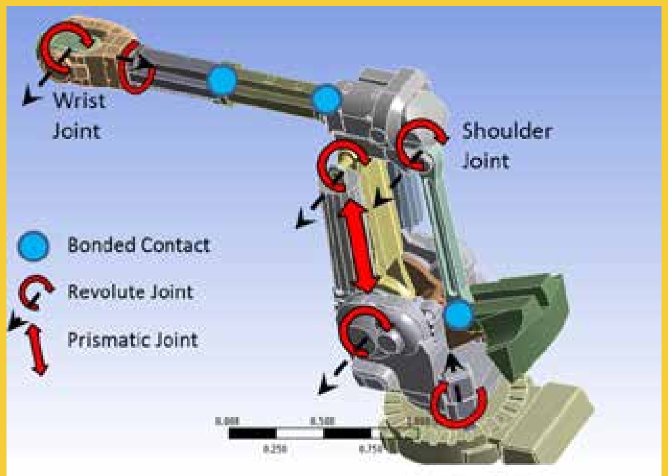


Figure 2 - Rigid Body Dynamics using rigid and flexible bodies

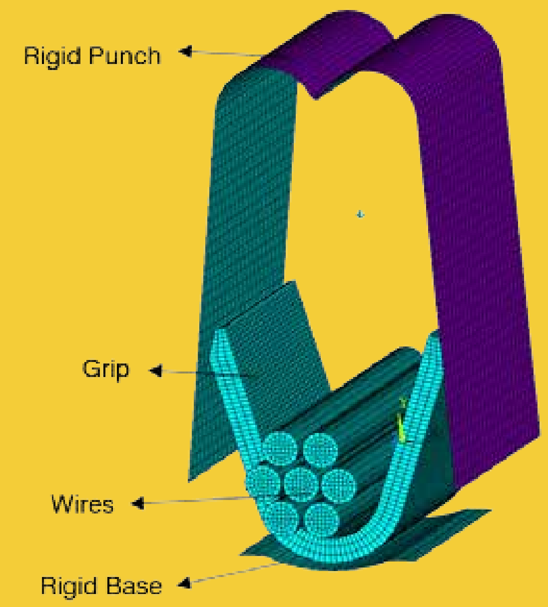


Figure 1 - New general contact in ANSYS 17

In the Mechanical environment it's now possible to have a direct access to the RBD (Rigid Body Dynamics) for cinematic and dynamic simulation of complex systems. It allows to simulate the cinematic behavior of systems with rigid and flexible bodies, exploiting the mechanical connections through joints and condensed meshes. The analysis of vibrations and noise is faster thanks to the improvements of parallel calculation and the implementation of new ANSYS ACT app, which helps to simulate the generation, propagation, radiation, absorption and reflection of acoustic pressure waves.

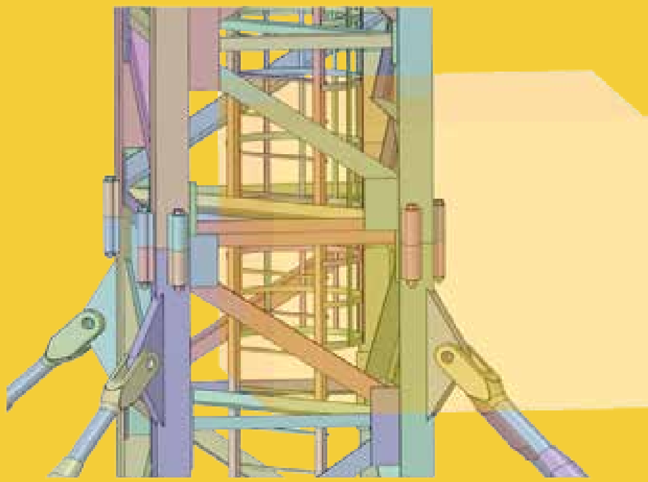


Figure 3 - Automatic volume extraction in SpaceClaim for Submodeling

New material models for geo-mechanical analysis have been implemented, to better simulate the presence of foundations, tunnels, compacted and consolidated soils. These models, which allow to introduce porosity effects, can be used also to simulate biological and anatomical parts, such as tissues and muscles.

Considering the tests of the seals watertight or, more generally, of an hyperelastic material introduced inside a gap between two components, it is possible to simulate the seal insertion, although it implicates very high deformations to the mesh: in fact, thanks to the "Mesh Nonlinear Adaptivity" method, the excessive elements' deformation is managed through the local re-meshing on the most critical areas. This methodology, applicable for 2D/3D elements, allows to simulate the assembly operation, granting an efficient numerical convergence.

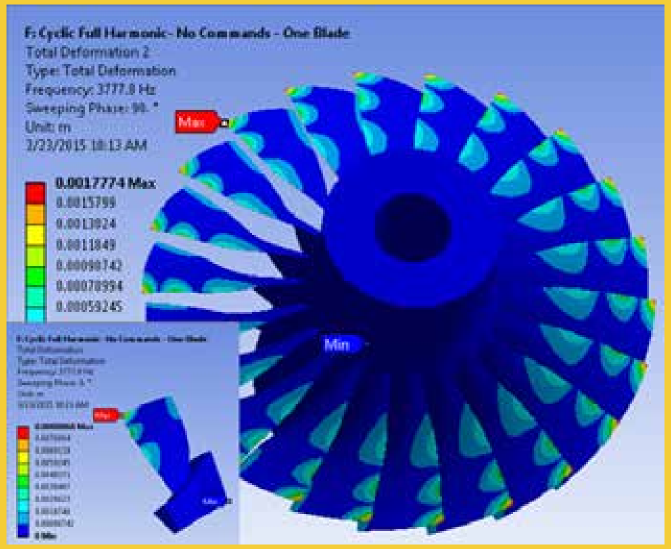


Figure 4 - CPU time reduction thanks to cyclic harmonic analysis available in R17

A specific application for the Offshore industry has been introduced to define a random waves spectrum in which energy is developed in different directions. Furthermore, the understanding of the physical behavior in the presence of these wave spectra has been considerably improved thanks to the introduction of new graphics and polar diagrams. Further improvements have been introduced in the interaction between moorings and seabed as in the management of the contacts between moorings and boat and eventually in the definition of interfaces with friction.

HPC turns out to be more efficient, stable and scalable with a high improvement of the solution speed particularly for the dynamic analysis (e.g. modal, structural transients, Power Spectrum Density, acoustic).

Starting from ANSYS Composite Prep/Post it has been developed the new environment "ANSYS Composite Cure Simulation" which allows to simulate the curing of composite laminates, thus providing residual strain due to stresses coupling effects for non-symmetrical and unbalanced laminates. A great solution to reach tight and robust tolerances.

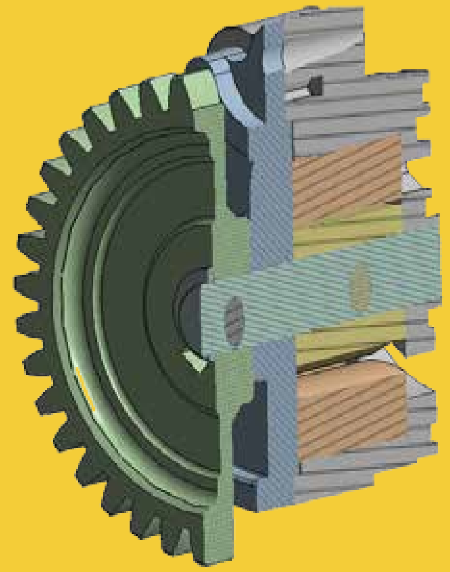


Figure 5 - Graphic enhancements in ANSYS Mechanical environments

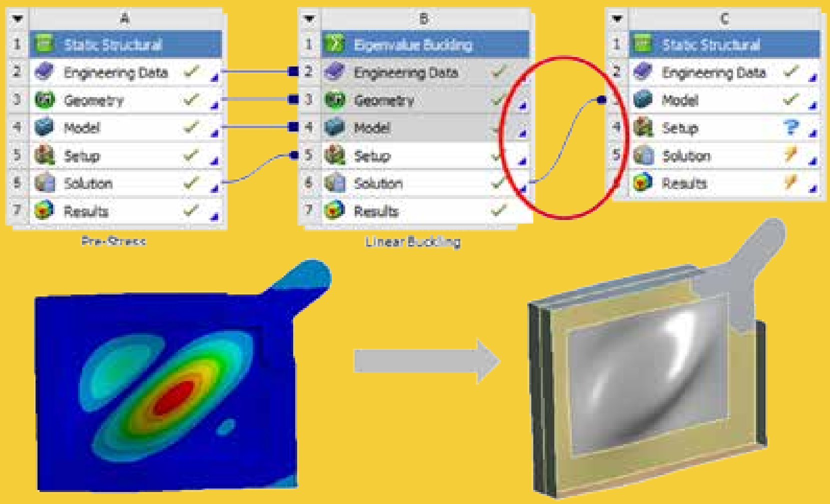


Figure 6 - Automatic export of deformed mesh and geometry using links in ANSYS WB

In the new age of APP and social networks sharing, even ANSYS goes in the same direction: the chance to develop custom models and for own simulation processes are no longer accessible only to software developers. Thanks to ANSYS ACT suite every single user has now the possibility to develop own new dedicated tools or use the already developed apps available on the ANSYS APP Store.

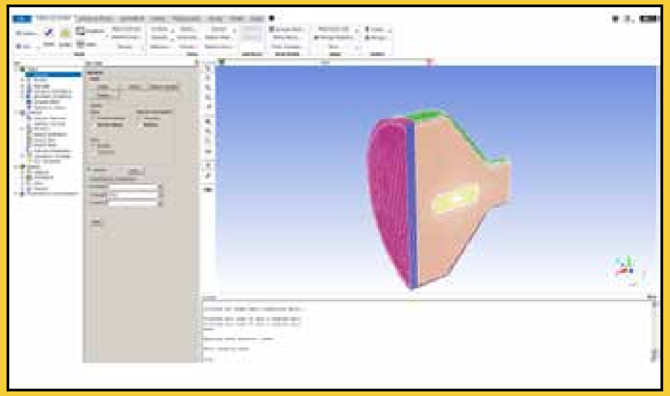
For more information:
Fabio Rossetti, EnginSoft
f.rossetti@enginsoft.it

ANSYS FLUENT 17.0

Ansys announces the new release of FLUENT R 17 with new capabilities and enhancements for a more comprehensive approach for the product design. The new features are described below.

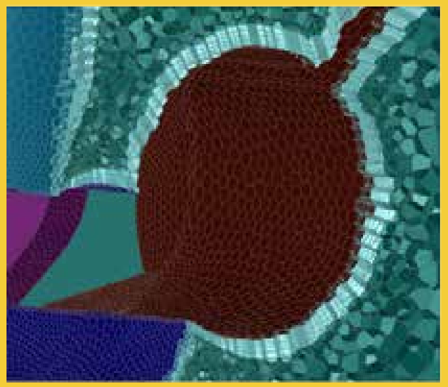
Fluent User Interface

The graphical interface consists now of an additional ribbon where the tabs are named, ordered and populated by the workflow. New button size and icons can be used for the common and most important commands with a significant reduction in the time needed for the setup of the problem. This layout is mainly focus on the primary fundamental workflow while keeping the secondary workflow layer like dialog boxes. The toolbar, the tree and the console (TUI, text user interface) are still available.



Fluent Meshing

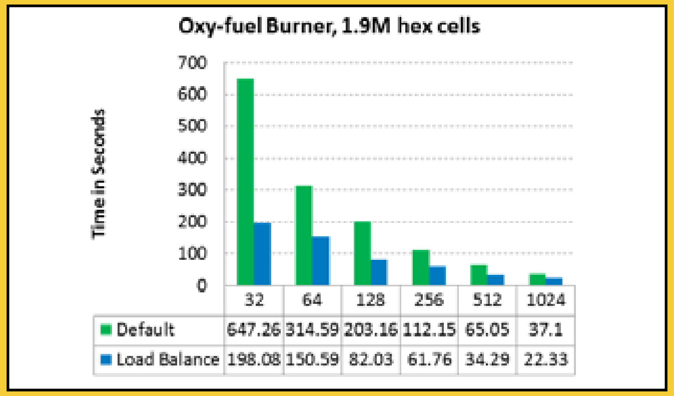
A cleaner design and better icons characterize the new version of Fluent Meshing. A CAD assembly mode can now be used to represent the CAD tree in the same way it is presented in the CAD package it was created with all sub-assembly levels. New algorithms for join, intersection and gap closing operations are available. An Auto Fill Volume function can be used for the region based meshing without the need to have the physical domain. Better performance are visible in parallel prism generation and the



polyhedral meshing is now native and 3 times faster compared to the tet generation and poly conversion in Fluent Solver.

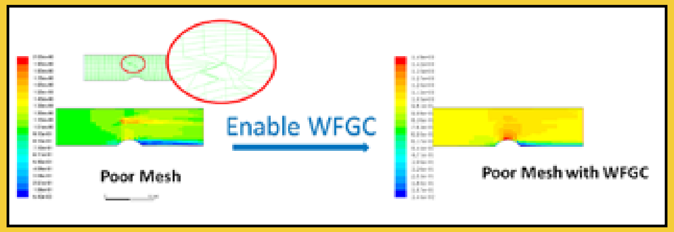
HPC

The new release R17.0 continues the development of the software with particular attention to the improvements in the HPC. Fluent has set the standard for many years in performance and scalability and with the new version an additional improvement in performance and scaling has been achieved. A new model weighted partitioning introduces a better load balancing between the flow and the physics, in particular with models like DPM and combustion.



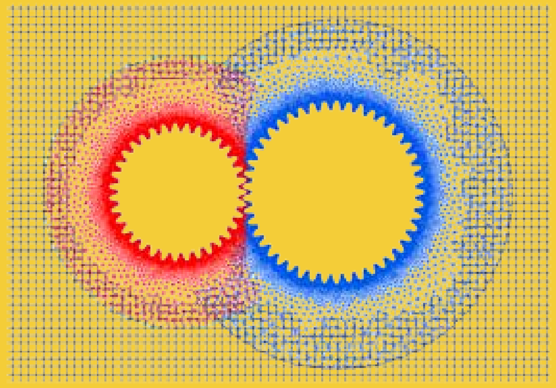
Physics

In Fluent R17.0 multiple changes have been done in order to improve convergence and robustness for typical industrial cases. The Conservative Coarsening method is now used by default and it is especially helpful for cases with native polyhedral meshes and highly stretched cells. A reordering is now performed within the AMG solver with benefit in the convergence speed. A new option in the computation of the gradient (Warped-Face Gradient Correction) has shown to be more resistant to error when the quality of the cell is very poor.



Fluent R17.0 introduces Report Definitions which are a step towards the unification of the monitoring and post-processing activities. The use of transient profiles for wall translation/rotation can now be used without the need of User-Defined function. A relative motion can be specified for rigid body and user-defined mesh motion in order to simplify the creation of the udfs. The Mesh Check includes different tools in diagnosing the quality of mapped mesh interfaces with the possibility to resolve the detected problems using the TUI. The overlap fraction can now be visualized in the contours.

The Overset Mesh method is introduced in R17.0. Such a

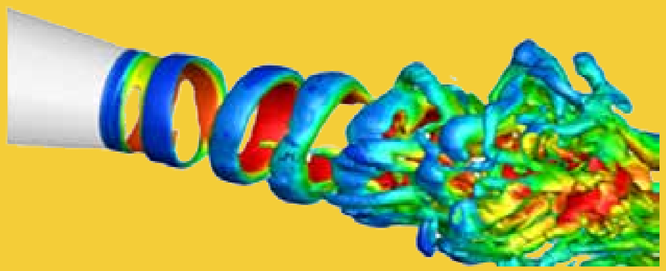


technology is a promising approach for cases where creating or remeshing a single mesh is impractical. Fully release capabilities support fixed-mesh steady and transient flow, single phase or VOF multiphase, heat transfer and turbulence. Moving mesh can be include as well by enabling Beta features.

New turbulence models are implemented, in particular for the Scale Resolving simulations. Two new options are Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES).

Since the acquisition of Reacting Flow, more functionality from Chemkin products have been integrated into Fluent. With the release R17.0 the Chemkin-CFD solver can now be used with no additional license and Fluent has full compatibility with all Chemkin mechanism. A Dynamic Cell Clustering with Chemik-CFD offers significant improvement for detailed chemistry.

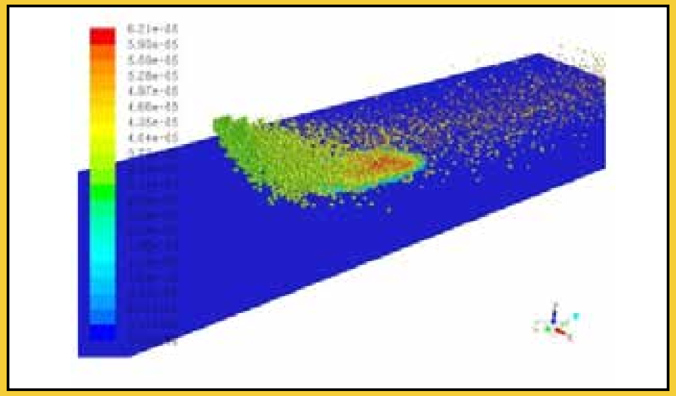
New capabilities for the Offshore/Marine applications include



the possibility to specify separate velocities for the primary and secondary phases and a multi-directional numerical beach can now be defined in order to improve the accuracy of results when multiple pressure-outlet boundary conditions are present. A new formulation of the wave theory has shown better accuracy and agreement with experiment.

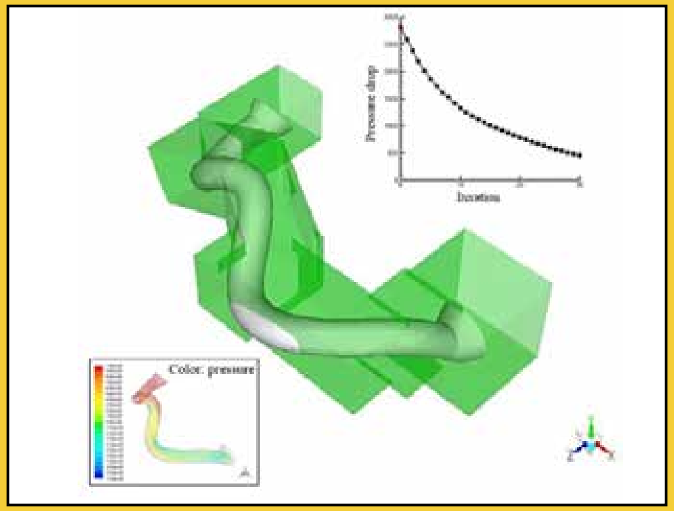
Rotation of particles is now a full release in the DPM framework. Additional DEM collision models have been implemented and the Macroscopic Particle Model is now included as an Add-on model in Fluent.

An improved formulation for the Lagrangian wall film can now be

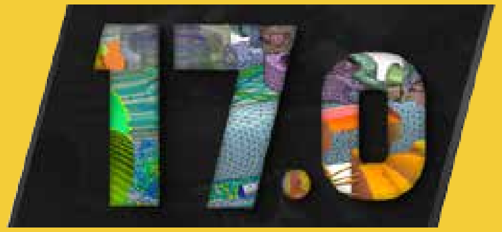


used and the Eulerian wall film now supports variable density of the film material.

Ansys Fluent R17.0 extends the capabilities of the adjoint solver. The compressible flow is now supported and different enhancements in the design conditions are available. The morphed shape can be constrained inside an arbitrary STL surface and prescribed motion (rotation, translation and scaling) can be applied as well. The Mesh Morpher/Optimizer tool benefits from an improved method for creating control points on boundaries.



For more information:
Michele Andreoli, EnginSoft
m.andreoli@enginsoft.it

ANSYS CFX 

ANSYS CFX announces new features and enhancements in order to allow engineers to improve their product's design. This brief article introduces the major changes to ANSYS CFX in Release 17.0.

NEW FEATURES AND ENHANCEMENTS

HPC

The time required to solve large simulations is still considered too long for typical industrial applications. This is why ANSYS continues to push very hard to enhance HPC capabilities. Main enhancements in R17.0 are described below.

Solution – Some examples

- Transient full hydro turbine (40M cells)
  - 10-30% reduction in solver time
  - 20% improved scalability at 512 cores
- External aerodynamics (100M cells)
  - ~40% faster on 4096 cores
  - Good scalability up to 25k nodes/core
- Engine internal flow application (380M cells)
  - >30% less solver time, scaling to 4096 cores
- 6 stage transient axial compressor (12M cells, 14 domains)
  - 20-30% reduction

I/O

Time to read and write files to HPC for large and complex cases with many regions/face sets could significantly lengthen overall solution time. So, optimization of CFX solver to HPC interface resulted in a substantial speed-up: I/O time now nearly negligible even at 64 cores.

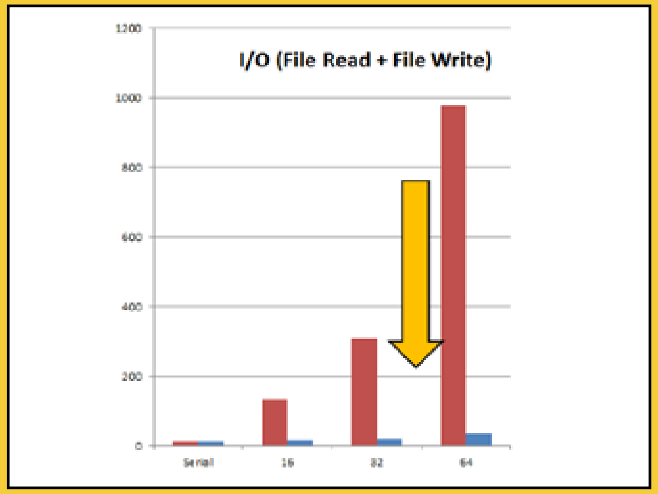
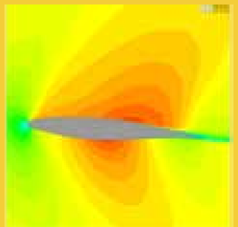


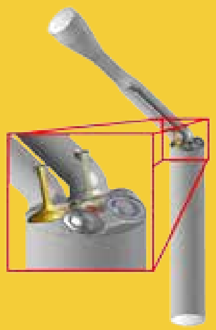
Figure 1: Reduction in wall clock seconds for I/O on an example test case with many regions

Turbulence

A new Intermittency transition model is available. It is a further development based on the two-equations Gamma Theta transition model. It basically reduces the computational effort by solving just one transport equation instead of two (only transport equation for the turbulence intermittency λ).

One of the weaknesses of the original DES model family is that it shields attached boundary layers insufficiently from the impact of the grid-dependent term. For this reason, the Stress-Blended Eddy Simulation (SBES) model has been developed. This model features a much improved shielding function to protect RANS boundary layers. The SBES model uses that shielding function, which is also utilized to switch to an existing algebraic LES model in the LES zone. The advantage is that you can clearly distinguish where the RANS and LES zones are (by visualizing the shielding function) and which model is used in which zone. Because SBES blends between two existing models, it is not a true turbulence model, but rather an automated way of switching between existing RANS and LES models. The SBES model "transitions" much quicker from RANS to Scale Resolving Simulation (SRS) mode in separating shear layers. This leads to more realistic solutions with higher internal consistency. In addition, this model allows for a RANS-LES "transition" on much coarser grids than classical DES.





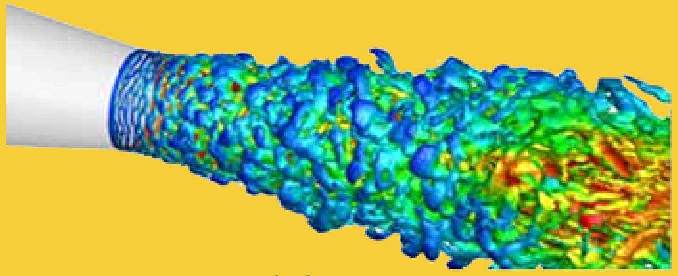
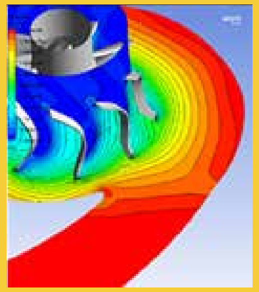


Figure 2: New turbulence model: SBES

Transient Blade Row

It is now possible to run a transient case that involves conjugate heat transfer.

For Time Transformation cases, the CFX-Solver Manager can display monitor points with a corrected (physical) time on any Plot Monitor; they no longer need to be displayed on a special "Time Corrected Monitor". In addition, users can export data from any Plot Monitor without any restrictions on exporting monitor points with different corrected time values.



A multi-disturbance gust analysis can be modeled now using the Fourier Transformation method. In this analysis, the blade is subjected to two simultaneous disturbances; one on the inlet and the other on the outlet of the blade passage. Typically, the inlet and outlet profiles are initially extracted from a steady-state stage simulation of a 1.5 stage configuration. The profiles are then imposed on the inlet and outlet of the rotor passage in a transient gust simulation. This modeling strategy can be used as an alternative for modeling a 1.5 stage compressor or turbine. It accounts for blade passing frequencies from the upstream and downstream rows on the modeled blades. The use of the Fourier Transformation method also allows for very large pitch variations such as with a fan subjected to inflow distortion.

Now the user can set up cases that involve a long-period disturbance using the Fourier Transformation Transient Rotor Stator (FT-TRS) interface. Some applications are:

- an impeller or rotor attached to a full 360° domain (vaneless or vaned volute)
- a fan in a crosswind

Multiple stage cases using the Time Transformation method can be set up.

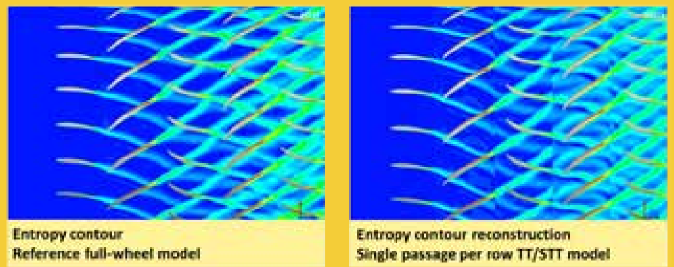


Figure 3: Traneint Blade Row – Multi-stage Time Transformation method

It is now possible to stop and then restart a Time Transformation or Fourier Transformation case with an increased number of time steps per period. In many flow simulations, it may be advantageous to start the simulation with a low number of time steps per period in order to pass through the initial transient phases of the solution, then later increase the number of time steps per period to accurately capture the flow physics. For a large simulation, this strategy can help reduce the overall simulation time required to reach convergence. You can stop a Time Transformation or Fourier Transformation simulation and then resume it with an increased number of time steps. An important constraint on how you increase the number of time steps for Fourier Transformation simulations is that the number must be doubled. For example you could start with 40 time steps per period, then restart with 80, then 160. There is no such constraint for Time Transformation simulations. For example, you could start with 40 time steps per period, then restart with 50, then 70.

System Coupling

Users can now use a System Coupling component system in ANSYS Workbench to perform coupled simulations that involve ANSYS CFX and ANSYS Mechanical. This technology was already available in the previous releases just for ANSYS Fluent. Users can set up a one-way or two-way fluid-structure interaction (FSI) design analysis in ANSYS Workbench simply by connecting a System Coupling component system to Mechanical system and to Fluid Flow (CFX) analysis system.

Participants that can be coupled with ANSYS CFX include:

- Static Structural
- Transient Structural

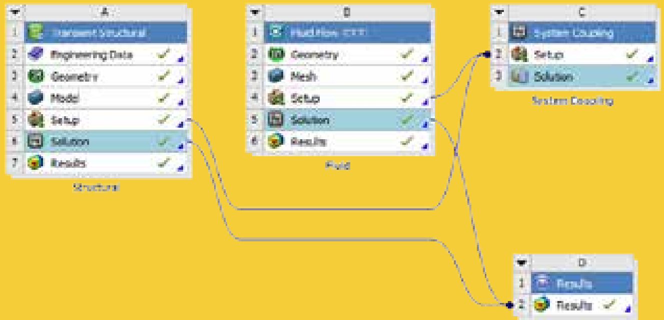


Figure 4: System Coupling example

Solution Caching for Design Point Studies

There are two new ways to control the retention of solution data:

For Old Design Point Solutions

- Solution cell property "Keep Latest Solution Data Only" (Properties View in the CFX Introduction)
- Shortcut menu command "Clear Old Solution Data" (Context Menu Commands in the CFX Introduction)
- Workbench preference "Keep Latest Solution Data Only" (CFX in the Workbench User's Guide)

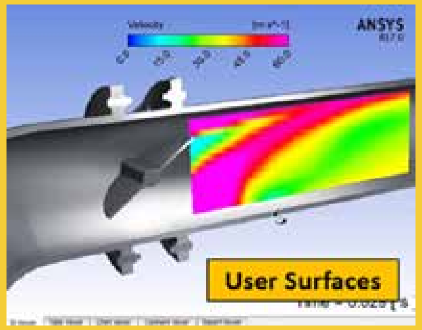
For Cached Design Point Solutions

- Solution cell property "Cache Solution Data" (Properties View in the CFX Introduction)
- Shortcut menu command "Clear Cached Solution Data" (Context Menu Commands in the CFX Introduction)
- Workbench preference "Cache Solution Data" (CFX in the Workbench User's Guide)

Large Problems

Large problem executables store integer numbers as 64-bit words. These executables allow larger mesh sizes than the default executables. A large problem executable requires twice the integer memory of a default executable, for a given problem and grid size. There is no advantage to using a large problem executable unless the problem size requires it.

Use in Partitioning and Interpolation



The default executable is currently limited to allocate 2^31-1 words of 4-byte integer stack space; this limits the maximum problem size for partitioning to approximately 80 million elements (structured) and 200 million elements (unstructured). Similar limits exist for interpolation.
Larger problems are likely to require the use of a large problem executable. In theory, a maximum problem size of two billion elements can be partitioned with this executable. However, practical considerations, such as available computer resources, will still limit the maximum size.

Use in Solution

The default solver is currently limited to a problem size of approximately 700 million nodes. Larger problems require the use of a large problem executable. In theory, a maximum problem size of two billion nodes can be accommodated.
As with partitioning and interpolation, the maximum size of a given parallel partition and the maximum problem size of serial solutions are limited in the default executables by an allocation limit of 2^31–1 words for any memory stack. This limitation does not apply to large problem executables.

User Fortran

You must have the Intel Fortran 15.0.2 compiler to build User Fortran for ANSYS CFX (Windows and Linux).

Variables

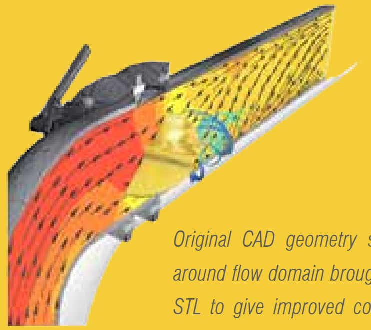
The following variables can now be selected for output into the results: `Spinodal Pressure`, `Spinodal Temperature`.

Enhancements to Stage (Mixing-Plane) model

- The `Constant Total Pressure` option for Downstream Velocity Constraint has been made the default option instead of `Stage Average Velocity`.
- Improvement to energy closure

These two changes improve the conservation of Total Pressure and Rothalpy across a mixing-plane. To revert back to previous numerics, use the following expert parameter option: `stage energy closure option = 1`.

Figure 5: Enhancements to Stage (Mixing-Plane) model allow to avoid spurious results across interface



Original CAD geometry surfaces around flow domain brought in as STL to give improved context to simulation results

CFD-Post

Transient problems produce very large volumes of data that fills up hard drives and must be processed by CFD-Post. Users may now specify arbitrary surface locations as output locations for transient data. Outputs only selected variables at selected frequencies on selected surfaces – and these surfaces can be completely independent of the mesh topology

- Reduces disk space enormously
- Much leaner, faster post-processing
- Fluent and CFX

Simulation images may be difficult to interpret without key identifying landmarks. In R17 it is possible to import/export STL format geometry definitions:

- Import as contextual geometry
- Export any surface locator for other uses

It is now possible to animate transient variations of flow solutions for CFX in a easy to use "Music player"-like controls:

- Play/stop
- Next/previous time step
- First/last time step

Response times are slowed by data compression steps between internal components of CFD-Post (Engine and GUI). Avoiding compression/decompression during data transfer allow to gain up to 40% speed-up in post-processing time.
CFX Monitor Data are available directly in CFD-Post Charts.

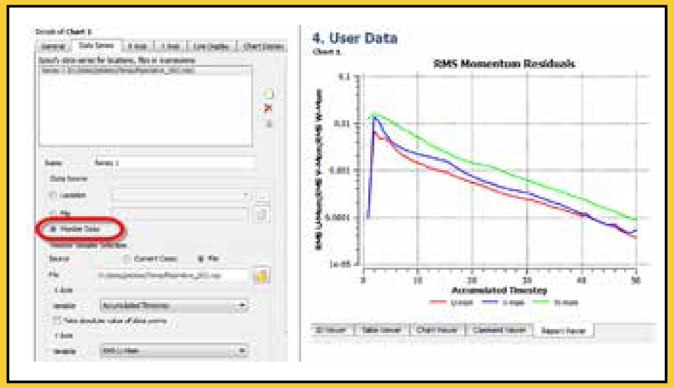
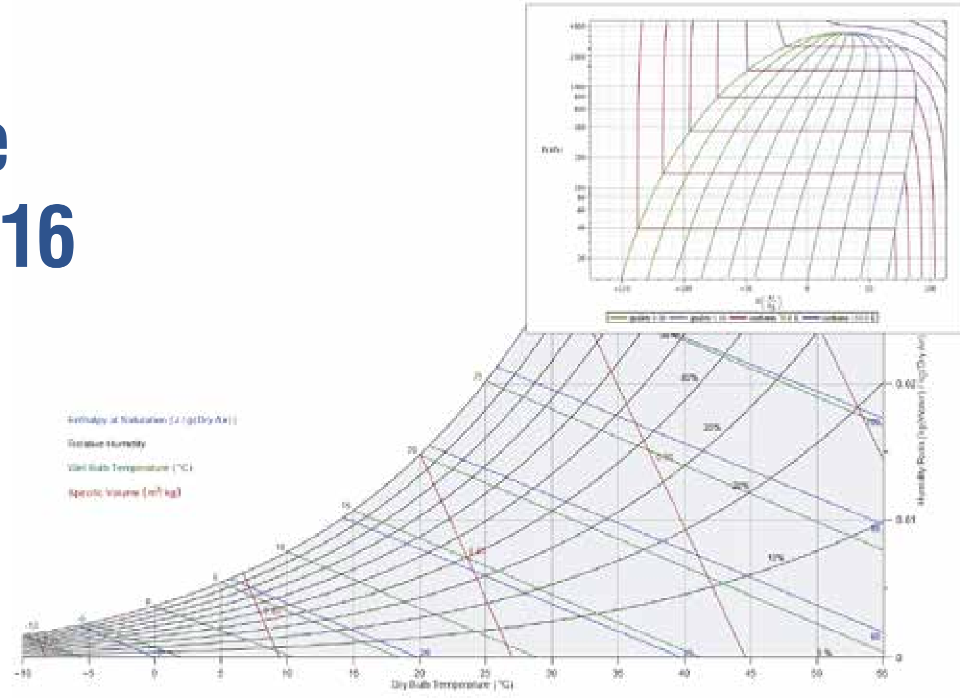


Figure 6: CFX Monitor Data are available directly in CFD-Post

For more information:
Alessandro Marini, EnginSoft
a.marini@enginsoft.it

Engineer more with Maple 2016 & MapleSim!



Maple 2016 key features for Engineering

The new Maple 2016 release carries many new features related to user experience improvements, productivity, efficiency and innovative capabilities. In this article we will highlight the novelties whose impact on engineering activities could be most effective.

- Perform calculations with thermophysical properties of pure fluids, humid air and mixtures, generate customized psychrometric charts, and more.
- Use new statistical analysis and visualization tools to gain insight into your data, including principal component analysis, heat maps and more.
- Code generation tools provide the ability to generate and export code in different languages or linkable libraries starting from Maple functions and expressions.
- Leverage new high performance computing tools that analyze your code to detect barriers to safe parallelization and help you resolve them.

Thermophysical Data

Starting from the 2016 release, Maple ships with a package containing a fluids database and the possibility to perform thermophysical properties computation.
Improvements on this topic include: pure and pseudo-pure fluid equations of state and transport properties for 118 components, mixtures of properties using high-accuracy Helmholtz energy formulations, correlations of properties of incompressible fluids and brines, high accuracy psychrometric procedures and charts, and automatic unit conversions.
For example, tools in this package give users a good starting point and extensive support to analysis and simulation of Organic Rankine Cycles; data obtained in such ways can then be combined with MapleSim Hydraulics database blocks in order model the hydraulic system and predict its behavior.

Data Analysis and Statistics

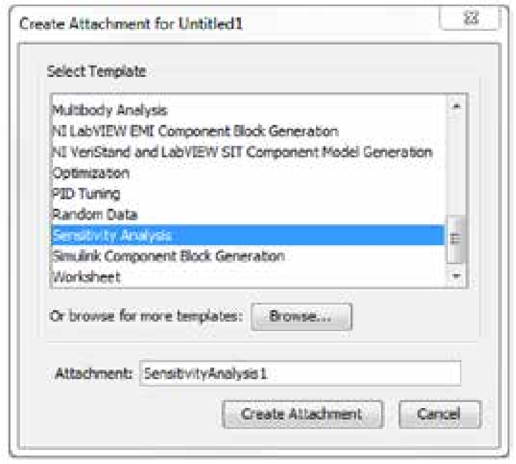
Improvements in the new Maple 2016 release also include data analysis and statistics features, for example new computation and visualization tools for principal component analysis.
Using Maple, you can determine which variables explain the majority of the variability in your data, and visualize the variance contributed by each of these principal components: this technique is especially useful when employed to recognize patterns and detect anomalies.

Maplesoft
Mathematics • Modeling • Simulation
A Cybernet Group Company

It is always useful to remind that the Maple suite can easily interact with a wide array of data formats, such as Excel files: this way, Excel worksheets can be used as data sources to produce fitting models that can also be integrated in MapleSim models.
Such models can then be studied by applying one of the many analysis templates in the MapleSim library, such as Monte Carlo Analysis, Sensitivity Analysis, and more.

Code generation

The CodeGeneration package is a collection of commands and subpackages that enable the translation of Maple code to other programming languages (C, Fortran, Java, Matlab, R, Visual Basic, …).
Optimized code generation produces high-performance, royalty-free code suitable even for repeated optimization runs and complex real-time simulations, including hardware-in-the-loop (HIL) applications. Detailed



benchmark results (summarized on the table) are available at the address
http://www.maplesoft.com/products/maplesim/symbolic_computation.aspx#4
MapleSim code generation tools convert system equations, previously simplified by Maple, to source code while applying symbolic optimization steps that dramatically speed up execution times. By removing expensive calculations from inside iteration loops, MapleSim can decrease the number of calculations for a single common subexpression from thousands to one in a typical application.
Available code generation targets, using MapleSim or MapleSim with a connectivity add-on, include:

- Standalone C code
- Simulink®/Simulink® Coder™
- LabVIEW™ and NI VeriStand™
- dSPACE®(DS1104 controller board)
- B&R Automation Studio
- VI-CarRealTime™
- Functional Mockup Interface (FMI)
- Maple: You can speed up your analysis computations by running compiled MapleSim models from your analysis documents

		Computational Cost			Differences
	Operation	Initial	Simplified	Symbolic Optimized	# Operations Removed
Active Suspension	functions	91	31	5	86
	additions	23	7	3	20
	multiplications	35	13	5	30
Vehicle Model	functions	708,738	390,465	80	708,658
	additions	110,168	34,690	8,362	101,806
	multiplications	318,479	232,161	8,942	309,537
Battery Start-up	functions	1,380	971	83	1,297
	additions	454	237	126	328
	multiplications	934	513	116	818
Ackerman Steering	functions	2,956	1,447	20	2,936
	additions	814	260	75	739
	multiplications	1,389	733	86	1,303
Space Rover	functions	2,726,936	329,575	90	2,726,846
	additions	426,755	33,287	6,386	420,369
	multiplications	1,048,636	257,072	7,431	1,041,205

The resulting code can be seamlessly incorporated into popular real-time toolchains and other applications, royalty-free.

Parallelism
Maple provides a parallel mode for its math engine, allowing it to evaluate multiple expressions at the same time when executed on a computer with multiple CPUs, therefore reducing overall computation times. Unlike many other commercial software, a normal Maple/MapleSim license will be enough to take advantage of all cores available on the machine.
For larger scale problems, even parallel mode on a single machine might not be enough to handle computations: this is where distributed systems come into action, and the Maple Grid Computing Toolbox makes it easy to create and test parallel distributed programs. Models can be created and fully tested using the local implementation inside Maple/MapleSim, and then deployed to the full cluster using the toolbox, without changes to the code.

Further reading
For more examples on MapleSim modeling capabilities, please visit the MapleSim model gallery:
http://www.maplesoft.com/products/maplesim/ModelGallery/
For further information about MapleSoft products:
Manolo Venturin, EnginSoft - m.venturin@enginsoft.it

ESTECO Enterprise Suite | EES combines the optimization technology of modeFRONTIER with SOMO, the enterprise collaboration and distributed execution framework aimed at managing the complexity of running multidisciplinary design projects

Globalization, web technologies and increasing product complexity have resulted in companies radically changing their approach to product design development and processes. Bigger, geographically distributed design teams specialized in different disciplines need to collaborate in order to get the job done. EES helps companies handle the complex process of managing and running multidisciplinary simulations. By extending the familiar modeFRONTIER desktop paradigm to the web-based collaborative environment SOMO, ESS empowers design teams with a sharing platform for models, workflows, simulations, optimizations and results. SOMO, a next-generation collaborative solution, brings global design teams together in a secure web-based virtual environment that enables teams to collaborate effectively across geographies, business units and skills. EES encourages collaboration by enabling design teams to share simulation and optimization data, models and strategies via a common shared repository that can be accessed anywhere, anytime on any device via a web browser, streamlining and speeding up the design cycle, be it for simple design problems or complex Multidisciplinary Design Optimization (MDO) scenarios. Design teams can upload and organize models and data, create and reuse multiple DOE and optimization strategies, execute huge numbers of jobs on a distributed execution network interfacing with HPC systems and cloud environments and perform data analysis using an array of post processing tools, safe in the knowledge that project integrity is safeguarded by reliable project versioning that aggregates design changes continuously under controlled conditions.

Somo is produced by ESTECO Spa and is supported in Europe by EnginSoft.
For more information: Francesco Franchini, EnginSoft - f.franchini@enginsoft.it

Flowmaster V7.9.4

Flowmaster V7.9.4 has seen a particular focus on usability. To this end a number of individual improvements have been included which, taken together, significantly improve the user experience. In addition, enhancements on physical modelling and connectivity to third party softwares are also introduced at this version.

User Experience
Persistent Run Button
In this new version the run button is now available from all tabs on the network view pane. The user will no longer need to go to the simulation data tab in order to run a simulation. This feature is of particular value when considered alongside the re-usable chart feature and changes to the experiments tab discussed below (see Fig. 1).

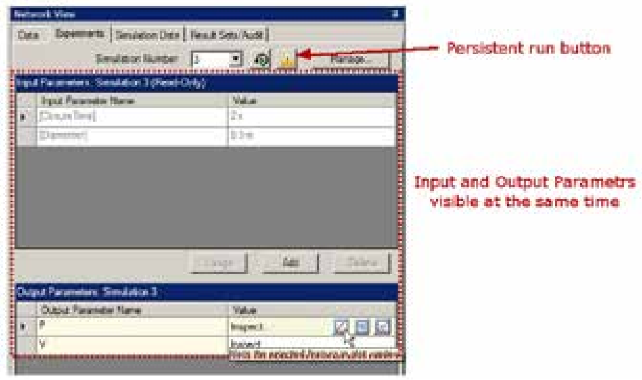


Figure1 - Persistent run button and visualisation of input and output parameters

The Experiments Tab and Parameters
The changes to the experiments tab and input/output parameters have been made with one overarching goal: to make it easier for users to modify key system parameters and observe the impact as quickly and smoothly as possible.
Firstly, in terms of creating parameters; users will now be able to create output parameters via a right-click on any relevant result feature. Output parameters will now be visible below input parameters, but on the same pane. By default, the latest result will be auto-selected, but it is possible to cycle through results via a drop down network at the top of the tab. The corresponding input parameter(s) will also be displayed. Both steady state and transient time histories are now available from the experiments tab in the same manner found on the current data tab. Output parameter results are also now available for non-Experiments simulations (see Fig. 1).

Bookmarks
In keeping with the ethos described above, a new "Bookmark" feature has been added to the charts window in Flowmaster. Bookmarks allows charts to be updated automatically on completion of a successful solution. It is possible to set references which persist in the plot, in order to allow users to compare the effects of a given modification with a benchmark or reference. It is possible to have more than one result feature per bookmark plot, and many bookmark plots concurrently in order to allow as detailed picture of a network as is required to be created (see Fig. 2).

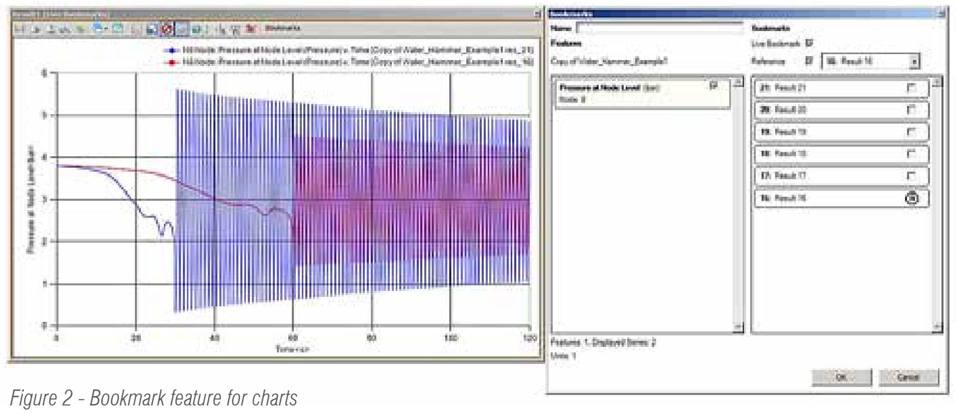


Figure 2 - Bookmark feature for charts

Filter Catalogue and Project Views
A filter box at the top of each view allows users to find all items that contain the character string entered in the box. It is possible to interact with any found entities at this point (see Fig. 3).

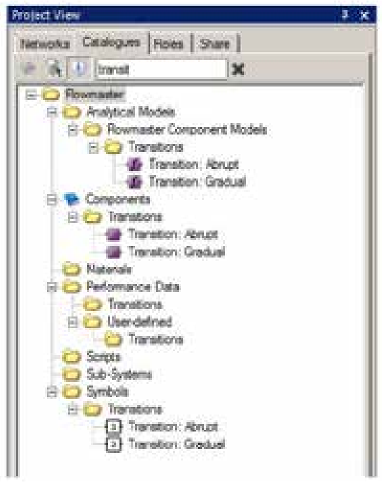


Figure 3 - Use of a filter in the catalogues view

Multi-Select in Catalogues

It is now possible to multi-select items within a catalogue using the standard cntrl or shift keys as required. This allows for more convenient copying, moving, deleting etc (see Fig. 4).

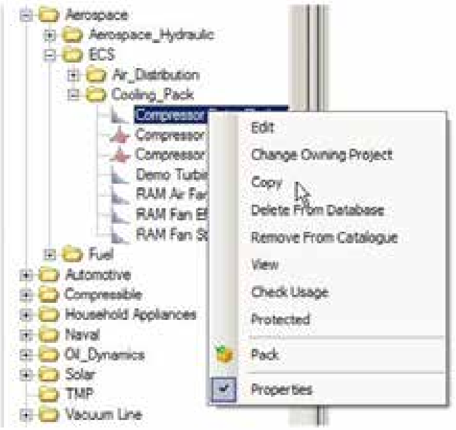


Figure 4 - Multi selection of items

Physics

Multi-Arm Tank

The multi-arm tank component has undergone a significant revision at V7.9.4. In general terms, the numerical stability of the component has been improved and the energy balance of the model has been revised with two distinct heat transfer models being implemented: polytropic and full heat transfer.

Cavitation

The implantation of this important feature was reviewed to ensure that it was being applied both consistently and in a manner that wouldn't compromise the stability of user systems. To this end, the artificial clipping of static pressure should it drop below zero was removed, although users will be warned if this does occur. The calculation of nodal static pressure – which will be compared with vapor pressure – will now be based upon the maximum dynamic pressure connected to the node. This approach is consistently applied should a cavity form.

NIST REFPROP Access

The NIST REFPROP fluid property database underpins Flowmaster's compressible and two-phase solvers. More information is now available regarding the accuracy of curves and surfaces generated by the user in Flowmaster. Should a generated fluid fall outside the tolerances specified, the user will be warned at the point of saving the fluid and advice given on how to improve the situation will be displayed.

Connectivity

Hydrodynamic Force Calculation and Export

It is now possible to calculate the hydrodynamic force generated by a fluid transient event and export the resulting force time history to different stress analysis softwares (SST CAEPIPE, Integraph® CAESAR II®, TRIFLEX and ROHR2).
The hydrodynamic force result feature is available for pipes, valves and transition components when run as part of an incompressible transient simulation (see Fig. 5).

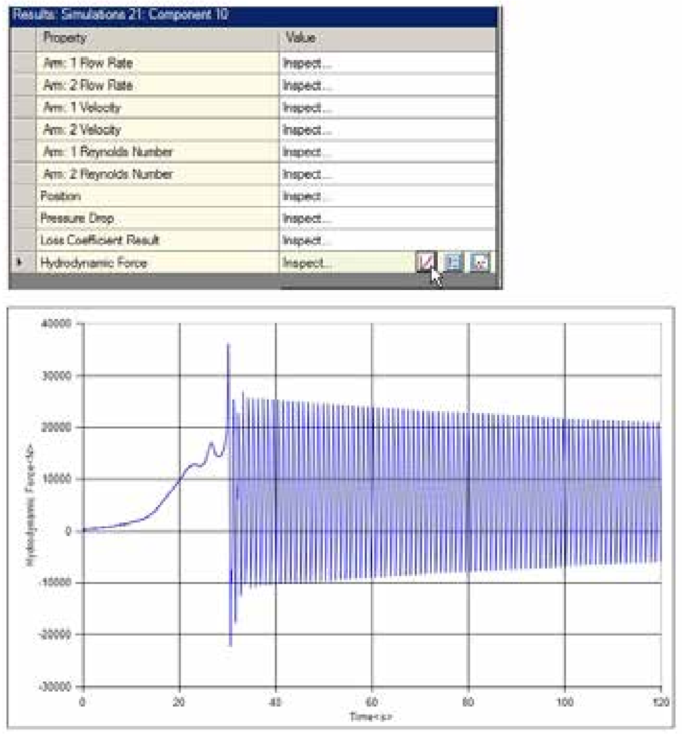


Figure 5 - Hydrodynamic force on a pipe during a water hammer

Flowmaster has been the tool of choice for 1D thermo-fluid simulation of pipework networks and systems for more than 20 years. Based on Don Miller's "Internal Flow Systems," Flowmaster is unrivaled in terms of accuracy, flexibility and cross-industry appeal due to its superior transient solver. Flowmaster delivers results you can trust from the earliest stages of the design process.
Flowmaster is supported in Europe by EnginSoft.
For more information: Alberto Deponti, EnginSoft
a.deponti@enginsoft.it

Lighter mechanical components using mesh morphing

Shape optimization is increasingly adopted in structural design. It allows to finely tune the parts getting products with better performances: accurate control of the stress level whilst keeping at the minimum the material usage and the weight. Reliable tools for addressing this discipline are one of the major needs of the industry fitting very well the modern trend of introducing additive manufacturing as a mainstream technology. An effective approach is provided by the use of mesh morphing as demonstrated in this paper adopting the FEA tool ANSYS Mechanical in conjunction with the ACT Extension RBF Morph.

Introduction

Shape optimization of structures is a well established topic, nevertheless its use in everyday structural design is emerging as a standard practice for the definition of a new component just in recent times. Among driving forces toward the massive use of shape optimization there are: the availability of High Performance Computing (powerful workstations and cloud resources), the high level of competition and the demanding market that needs better performances in less time, and the fast growing of new manufacturing processes based on 3D printing that allows the construction of complex shaped components not affordable for their complexity and cost when adopting standard processes.
Numerical tools based on Finite Element Analysis (FEA) are quickly growing to better fit aforementioned needs so that topological and shape optimizations are moving from the niche to the mainstream methods. One of the critical issues in the definition of automated optimization processes is the challenge posed by the efficient update of the computational grid; and this task becomes more and more complex because of the large nodes count reached when high fidelity models are used. It's quite common to use FEA models comprised of millions of nodes. Standard workflows rely on the generation of a new mesh for each shape variation which is updated according to the input parameters of the CAD. This is a very flexible approach that suffers of two main drawbacks: the parameterization of a complex CAD assembly could be quite a long and difficult task; the robustness is not that high as the regeneration of complex and unstructured meshes can fail and/or comes with the re-meshing noise (it means that the effect of the mesh dependence can be similar to the effect of varied parameter). New technologies based on direct modeling (as ANSYS Space Claim) or on mesh morphing allow to overtake the limitations of the standard process. Mesh morphing allows to update the mesh keeping exactly the same topology just updating nodal positions so that the new shape can be accommodated adapting the baseline mesh.

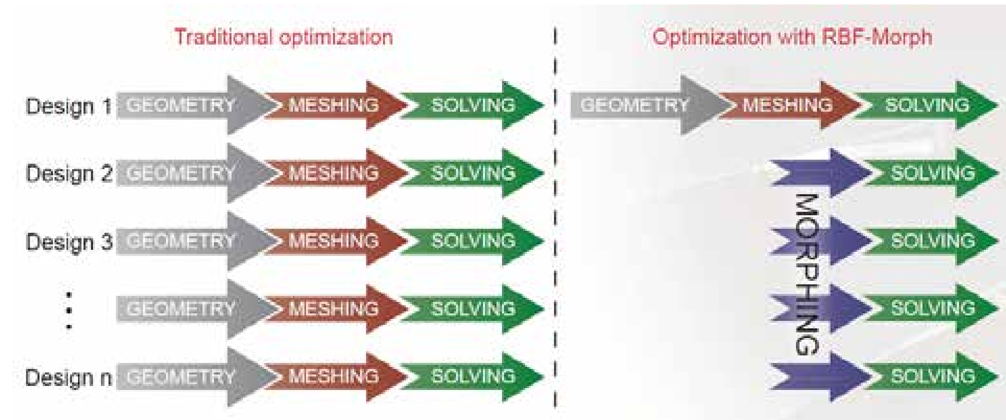


Figure 1- Comparison between a traditional optimization workflow and a mesh morphing based one

Mesh morphing as a shape parameterization tool

A comparison between a standard workflow and a mesh morphing based one is explained in Figure 1.

The applications proposed in this paper are developed exploiting the software RBF Morph ACT Extension which implements advanced mesh morphing based on the method of Radial Basis Functions (RBF) inside the FEM program ANSYS Mechanical. It's a new product released in 2015 and based on the technology of the ANSYS Fluent Add On available on the market since 2009 (www.rbf-morph.com). A study published by SACMI of Imola demonstrates how to integrate morphing and topological optimization tools. A further successful application by SACMI is demonstrated in Figure 2. Fatigue life optimization of a connecting rod of an internal combustion engine has been studied by the researchers of the University of Rome "Tor Vergata" and published at last AIAS congress. An effective application of mesh morphing has been adopted by Motocorse and is summarized in Figure 3 where the optimized shape of a motorbike racing part that combines style and performances is shown.

A typical workflow based on ANSYS Workbench is represented in Figure 4 where all the pieces required to set-up an optimization based on RBF Morph can be indentified: a parametric geometrical model (A), a structural analysis (B) that thanks to the morphing includes shape parameters, FEModeler (C) that is used to regenerate the final CAD of the optimal morphed shape, the parameters (Parameter Set), a DOE based response surface (D) and an optimizer linked to the response surface (E).

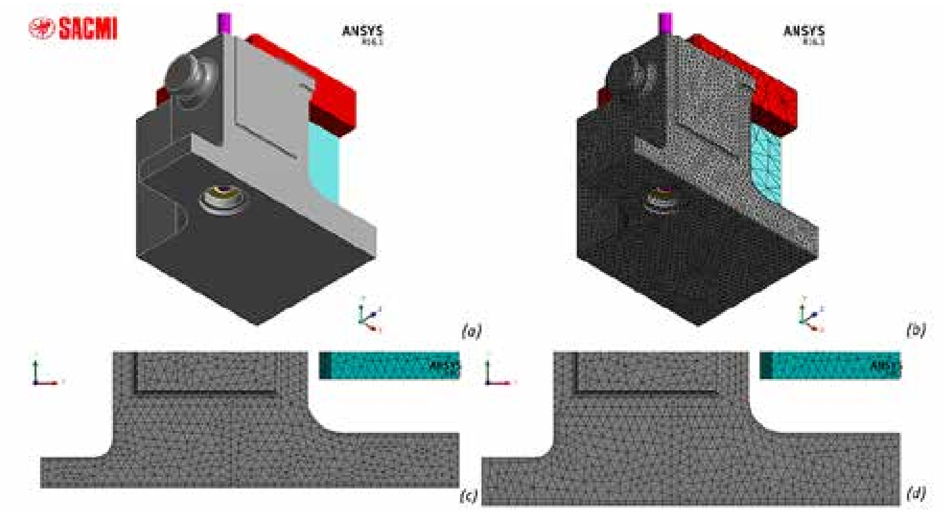


Figure 2 - Thanks to RBF Morph ACT Extension, SACMI performs mesh-based shape optimization within ANSYS Workbench overcoming parametric CAD-based shape optimization limits. In the example the thickness of the grey component is changed by preserving the contacts of the whole assembly

Figure 3 - The Republic of San Marino company Motocorse, with the support of the engineers A. Ridolfi and F. Giorgetti, has designed and optimized using ANSYS Mechanical and RBF Morph a CNC billet machined racing version of the rear suspension rocker for the motorbike Ducati 1199 Panigale

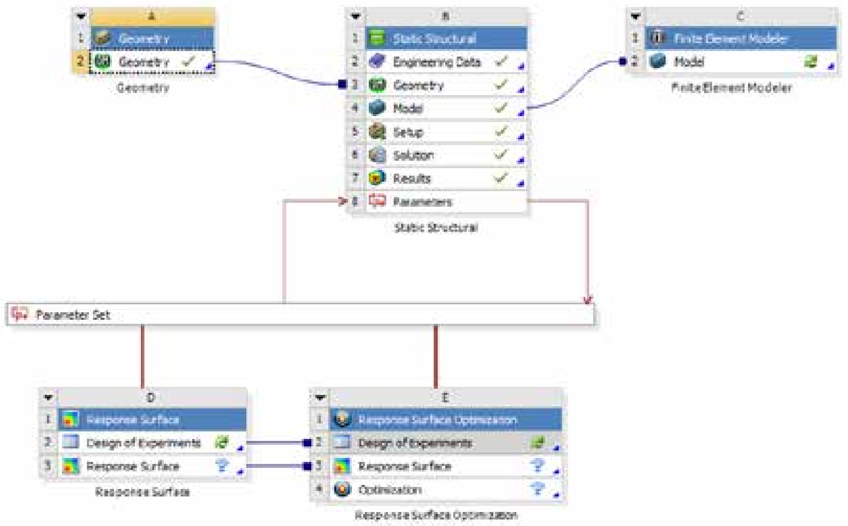


Figure 4 - Optimisation workflow in ANSYS Workbench. A geometrical model is used to generate the baseline (a static analysis in this example). Input parameters (that feed RBF Morph) and output parameters (computed by FEM solver) are steered by the optimization software DX (adopting in this case a response surface method). Optimal design point is sent to FEModeler that allows to generate the CAD of the new shape

Optimization of a wheel rim

To give a better insight about the proposed approach a detailed application, extracted from a paper published by D'Appolonia (RINA Group) in the proceedings of last CAE Conference in Italy, is given in this section and has as a target the reduction of the weight of a wheel rim while keeping the level of safety imposed by the testing standards applied in this field.

Baseline study foresees the modeling of the rim represented as a solid. The geometry is repeated cyclically eight times but boundary conditions are not cyclic and so the full model is required. All the loading conditions are simplified in static ones, and are defined with the aim to reproduce virtually the experimental testing cycle defined according to the TÜV standard. According to the standard, fatigue strength has to be considered and each of the load case produces as output a failure index that accounts for load repetitions and relevant strength assessment criteria.

The workflow defined in Workbench allows to solve all the load cases within the update of a single design point and retrieve as output all the failure indexes, the part mass and the part moment of inertia about the wheel axis. The rim is made in Aluminium alloy AlSi7Mg, material data are summarized in Table 1. Boundary conditions are sketched in Figure 5; the experimental testing cycle is conducted constraining the rim and loading the hub with an alternating tilting moment (two levels of intensity: ABM 50%, ABM 75%), an alternating torque (AT) and a vertical load representing the car weight acting on a single wheel (CW). For each loading conditions allowable fatigue strength are computed and then composed with resulting stress to obtain a failure index.

Property	E (MPa)	ν (-)	S_y (MPa)	S_u (MPa)	S_F (MPa)	ρ (kg m^{-3})
Value	$71 \cdot 10^3$	0.33	280	314	89	2770

Table 1 - Material data of the Aluminium alloy used for the manufacturing of the rim

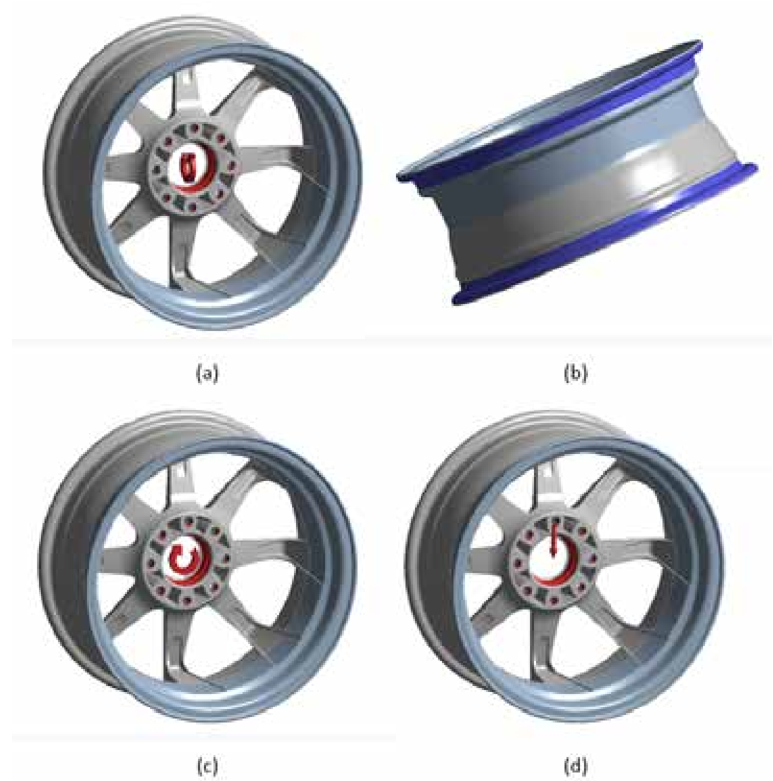


Figure 5 - The wheel is loaded using an alternate bending moment (a), an alternate torque (c), a load representing the car weight (d). Loads are introduced acting on the surfaces that connect the rim to the hub. The wheel is constrained at the interface with the tyre (b)

Mesh morphing is adopted to understand if a lighter shape of the spokes, obtained by a scaling operation of the spoke itself with respect to its transversal symmetry plane, is feasible. RBF Morph set-up foresees to control individually each spoke getting eight shape parameters. Such parameters are reduced to a single one imposing even constraint equations in Workbench.

Set-up is demonstrated in Figure 6 where it's clear how the morpher acts in the tree just after the mesh generation and Named Selections definition. In this example all the nodes of the mesh are used as target for the morphing action (wheel-rim); the shape is controlled acting individually on the shape of each spoke by a scaling operation (spoke1-scaling, spoke2-scaling,…,spoke8-scaling), keeping fixed the nodes of the hub and of the body of the rim (fixed-holes, fixed-external-surfaces) and leaving the remaining nodes free to be deformed by the morphing action adopting a linear RBF. A detail of how the nodes of a single spoke are controlled is represented in Figure 7 where the preview tool is used to check the correctness of set-up during its definition.

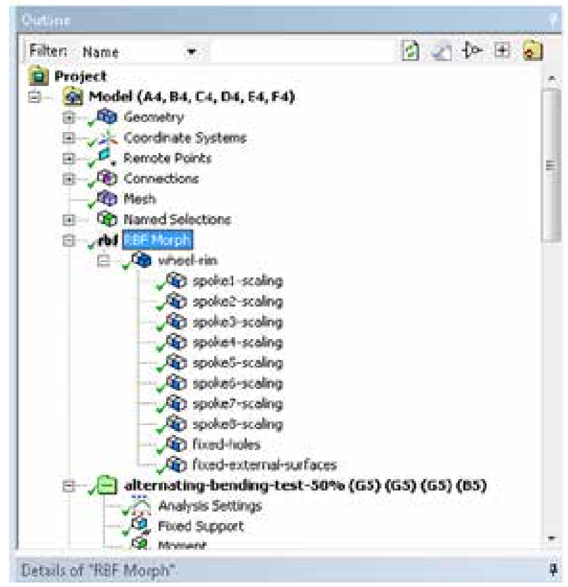


Figure 6 - A detail of the Tree in Mechanical; RBF Module acts just before the mesh generation and the definition of the Named Selections. It's comprised of a set of target nodes (wheel-rim), of 8 set of source nodes controlled by a scaling (spoke1-scaling, spoke2-scaling,…,spoke-8-scaling) and of two set of fixed nodes (fixed-holes, fixed-external-surfaces)

Table 2 summarizes the values of the parameter computed as a function of the shape parameters. It's worth to notice that the update of the Design Points with parametric shape is fully automated. According to the Workbench standard, new design variables (input and output) can be exposed in WB simply checking with a P the desired fields, including the input within the set up tree of RBF Morph. Such automation can be controlled by means of the optimizer that comes with Workbench, or by means of external software as modeFrontier in the study conducted by SACMI.

For the wheel rim studied the best material exploitation is obtained adopting a scale factor equal to 0.72 (Table 2) which leads to a weight reduction of about 0.5kg (-6.6%) whilst guarantying the safety levels required by the TUV standard. Stress levels of the original and the optimized solution are compared in Figure 8.

In this study a parametric CAD was not available and so the completion of the workflow requires the generation of a new one using a back2cad process. FE Modeler allows to get the geometry represented in Figure

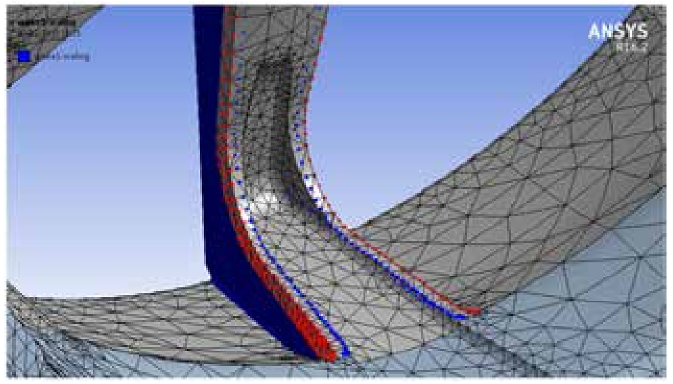


Figure 7 - Detail of RBF set-up used to control the shape of one of the spokes (spoke1-scaling); red points represent the node used as source points in the original position, blue points represent their preview in deformed position

TÜV test	ABM 50%	ABM 75%	AT	CW
FI scaling 0.8	0.75	0.91	0.33	0.31
FI scaling 0.7	0.83	1.19	0.41	0.39
FI scaling 0.72	0.82	0.99	0.39	0.34

Table 2 - Failure Indexes of the four load conditions are computed as a function of the scaling parameter that control the shape of the eight spokes

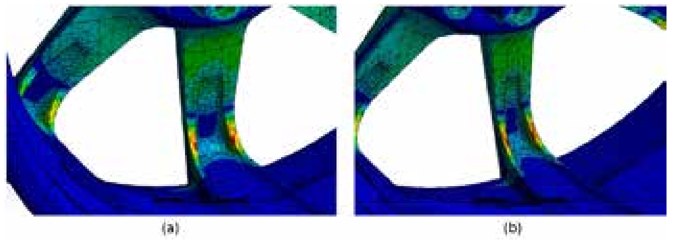


Figure 8 - Comparison of stress level between the original FEM model (a) and the optimized one (b); the new shape allows to reduce the weight getting a better exploitation of the material

9. It's important to highlight that such process is facilitated because the topology is preserved and surface reconstruction can be driven generating automatic selection which associates the new mesh with baseline geometry. The result is a new CAD that maintains the same topology of the original one (same face count).

Conclusions

Presented study shows how shape improvements of a component are feasible in a very short time adopting mesh morphing. The great flexibility of Workbench makes feasible several scenario that are relevant for the designer perspective. Validated legacy models (often available as dead meshes) can be updated to account for small updates decided during the service of the component without the need (and the high related cost) of generating a new mesh. Auxiliary CAD models of the area to be updated can be used as targets; a new shape can be easily introduced and if the target is parametric (for instance introducing control parameters in Design Modeler) an hybrid workflow, in which the geometry is controlled by the CAD and the mesh update by the morpher, is possible.

The morpher offers an high flexibility and the basic shape modifications (Translation, Rotation, Scaling, Curve Offset, Surface Offset, Curve Target, Surface Target) can be chained together to define new ones. For the introduction of new advanced feature the user can also customize RBF Morph with an ACT Extension that can override the displacement field of the source points contained in a node of the Tree. Such functionality has been used to drive the shape directly with mesh output: a surface can be sculpted according to stress levels; a crack can be extended according to local values of fracture parameters.

The great potential of the high performance RBF solver that is under the hood of RBF Morph software line is not yet fully discovered because of the novelty of the approach and the youngness of the product. It is worth to highlight that major steps toward the development of this technology have been driven by specific industrial needs. The developing team of RBF Morph is active in many research and industrial projects and is open to cooperate for overtaking new challenges.

Marco Evangelos Biancolini
University of Rome – Department of Enterprise Engineering "Mario Lucertini"

The new RBF Morph ACT Extension for ANSYS Mechanical is now available on the ANSYS App Store.

After the success gained within ANSYS market by the commercial mesh morphing solution for CFD models, RBF Morph ANSYS Fluent Add-on, RBF Morph team has developed a new software, based on the same technology, for the users of ANSYS Mechanical.

RBF Morph ACT Extension, developed with the Application Customization Toolkit (ACT) of ANSYS, is a software fully embedded in the GUI of ANSYS Mechanical and allows to make a model parametric just in a few clicks.

RBF Morph ACT Extension is distributed by RBF Morph (www.rbf-morph.com). A free trial, with limited features, is available for direct download on the ANSYS App Store.

For more information: Matteo Zanoletti, EnginSoft
m.zanoletti@enginsoft.it

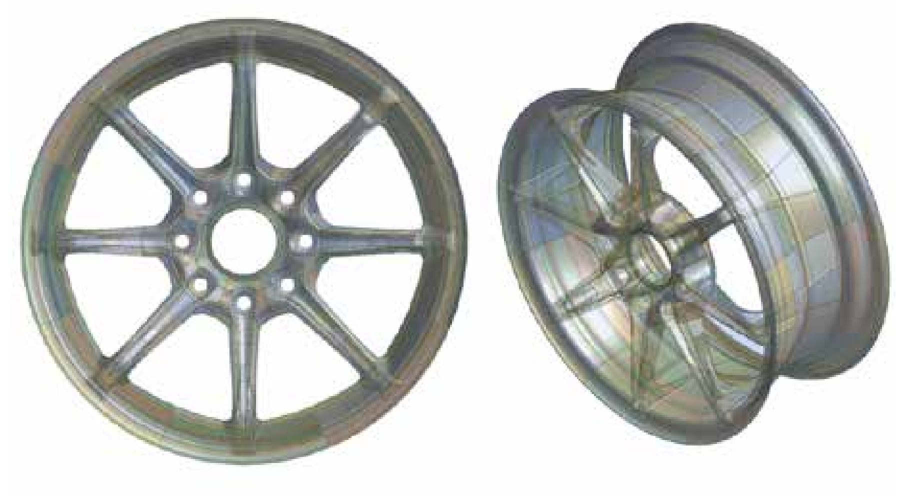
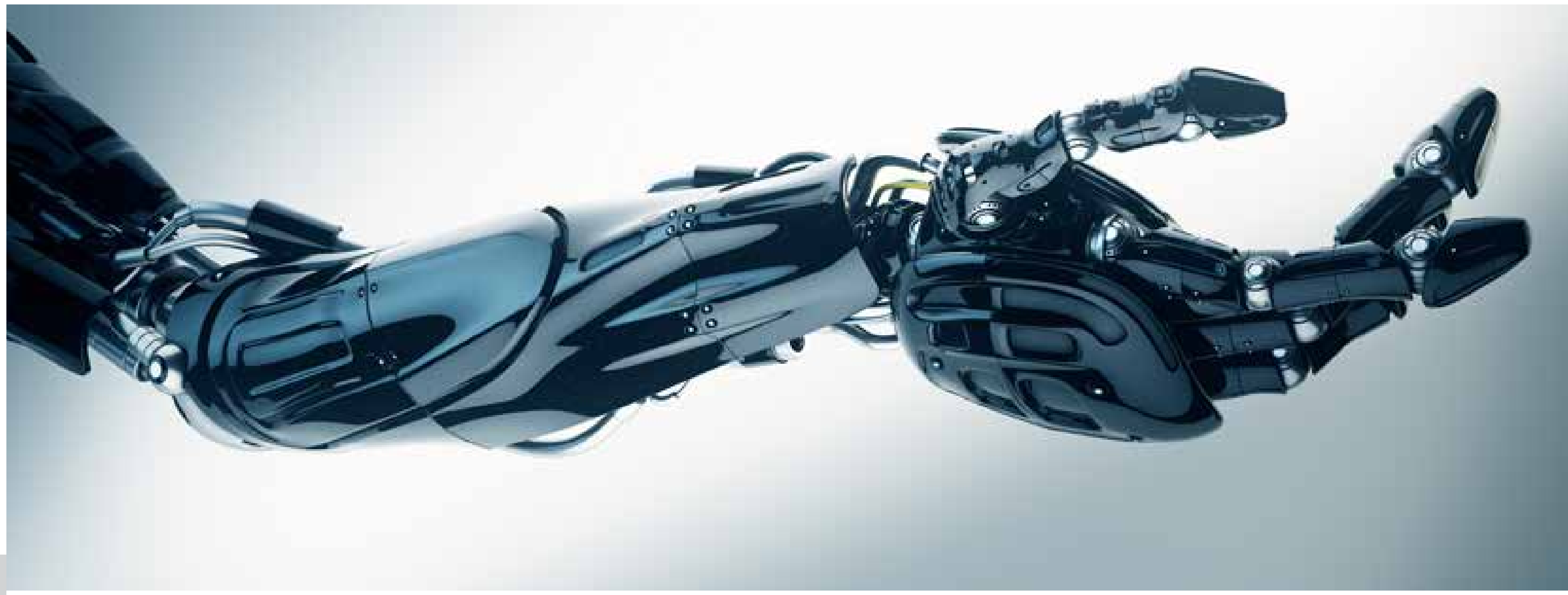


Figure 9 - The new CAD model obtained starting from the optimized mesh

Workcell Simulator: programming made easy



IT+Robotics | Spin-off of University of Padua

In the current context of small-medium business production, which is increasingly characterized by higher product personalization and small production lots, machine or work cell programming systems used in production play a key role in keeping prices competitive on the market. The use of advanced yet easy to use programming systems not only leads to a reduction in personnel training costs, but also to an effective reduction in the cost required to place a new product or variant on the market or simply to optimize the production process for a consolidated product. Furthermore, when programming via a simulator that recreates work cell behavior on the PC, another economic benefit is gained in not having to interrupt production to program a new product.

Bolstered by these considerations, IT+Robotics launched research in this field in 2007 which resulted in the release of the Workcell Simulator software solution. Workcell Simulator is the complete suite of applications for the simulation of the work cell, allowing programming in a 3D visual environment. Starting from the design of the piece to be machined, Workcell Simulator allows the entire work process to be programmed off-line. Using post processors, the program can thus be compiled to be used by the most popular manufacturers' robots and machinery. All programming occurs in a simulated environment, without having to occupy the plant during programming.

Workcell Simulator design is based on two goals: easy programming and solution flexibility.

Ease of programming is the key to reducing training time and optimizing programming efforts by the user. Various measures were taken to make programming easier and more effective. Firstly, the user interfaces were designed with special ergonomics, using familiar and easy to use paradigms. Various wizards were also introduced to allow the less expert user to complete software without the need of special training or external support.

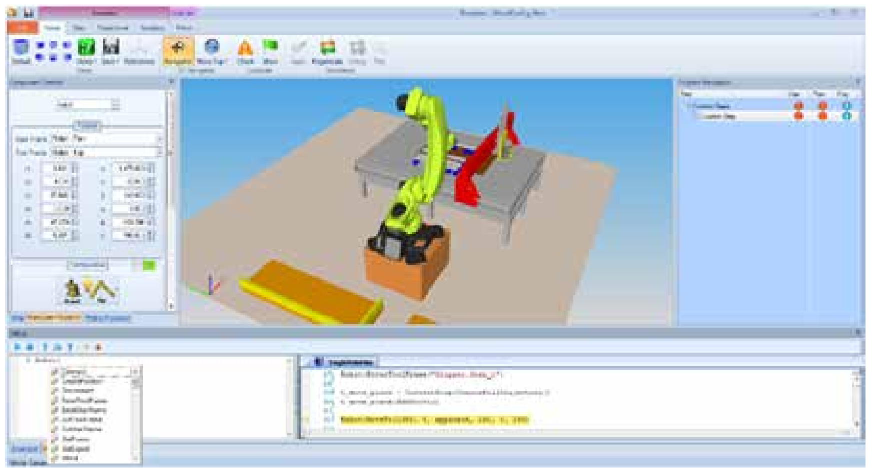


Figure 1 - Workcell Simulator used to program a wood cutting cell with a robot

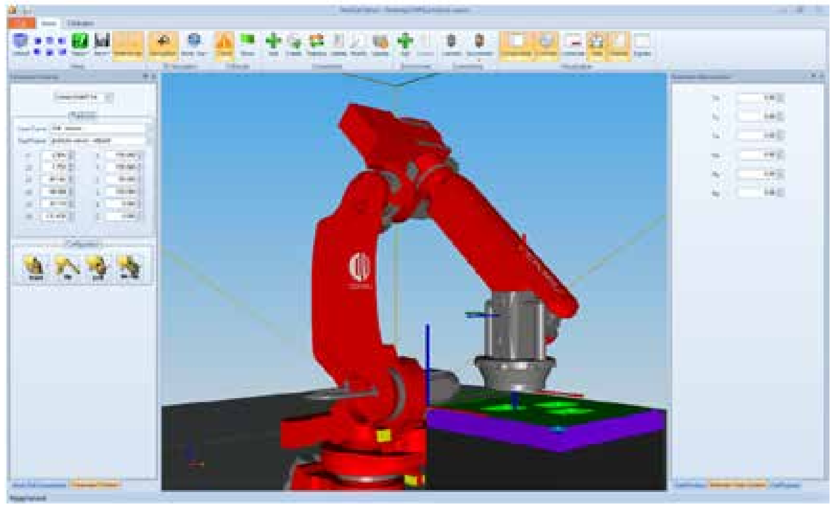


Figure 2 - Workcell Simulator used to set a quality control cycle in a cell with viewing system mounted on a robot

Another level of simplification was achieved by the addition of automatic calculation functions that allow the user to focus on what is deemed most important, leaving the details to the algorithms in the software. Automatic trajectory calculation functions are of special importance, especially applied to the robots. The algorithms added to the simulator, the result of years of research and association with the University of Padua, efficiently calculate the trajectory to move the robot to the points set by the user, automatically calculating the intermediate points to avoid obstacles in the work cell. This guarantees the absence of collisions during movements, while optimizing the cycle time. The same algorithms can also be used for multi-axis machines to calculate, for example, the release trajectory between one work phase and the next. The final step in the simplification process was creating a fully autonomous programming system that allows a new code to be introduced from a product CAD model with a single click. To achieve this results, advanced algorithms, called task planning algorithms, specifically designed to embed programming logic in a certain process, can be integrated in the simulator.

The second Workcell Simulator goal was to provide a single solution that can be adapted to any machine and process. To achieve this goal, Workcell Simulator was developed with an extremely modular approach: each single software behavior detail can be customized, starting from the logic of the components used to the features of the work cell, from the information collected by the user for programming to the generation of the code to be loaded in the controller. Thanks to the chosen approach, Workcell Simulator can be used to control even extremely different machines or processes without limiting solution potential.

IT+Robotics has followed various projects based on its Workcell Simulator product. Among the first verticalizations, off-line robot cell programming was considered in the sheet metal sector, specifically for press-bending. Workcell Simulator was used to create a software that allows all robot bending process details to be set, including the loading and palletizing phases, starting from the bent product 3D model. Another example is given by the verticalization able to permit off-line tube bending machinery programming. A significant application sector is visual quality control system programming based on robots. In this sector, research conducted by IT+Robotics, also due to three projects funded by the European Community (3D Complete, Thermobot and Fibremap), led to the development of Smart Check 3D, the IT+Robotics solution for flexible quality control based on the use of vision devices mounted on robots. Workcell Simulator was also used in this case as a basis for the plant programming software. The introduction of automatic trajectory calculation algorithms for the control cycle was of particular importance to permit the 100% coverage of the concerned product area, obtaining a solution that only requires a single click to introduce a new product code. This solution can also be used for other types of automated robotic processes such as, for example, painting or shot peening.

Thanks to its extremely flexible and versatile use, Workcell Simulator is the perfect solutions for machine manufacturers and system integrators interested in improving the features of their programming environments.

Learn more: http://www.it-robotics.it/products/3d-simulation/workcellsimulator/?lang=en
E-mail: info@it-robotics.it

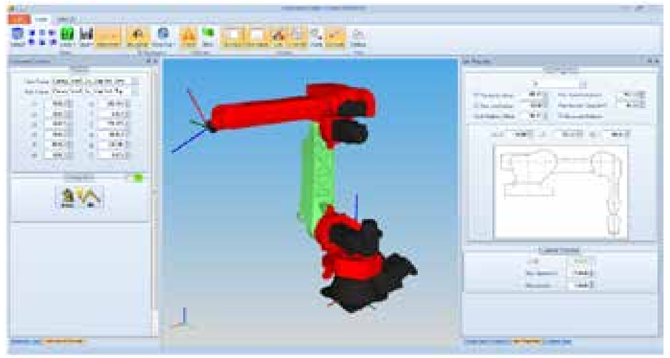


Figure 3 - Six-axis robot editor. It allows static and dynamic robot features to be customised

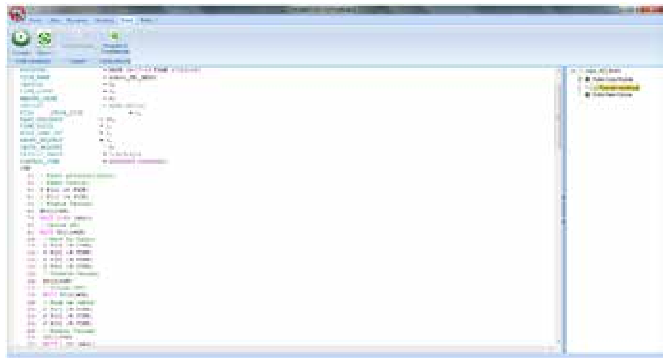
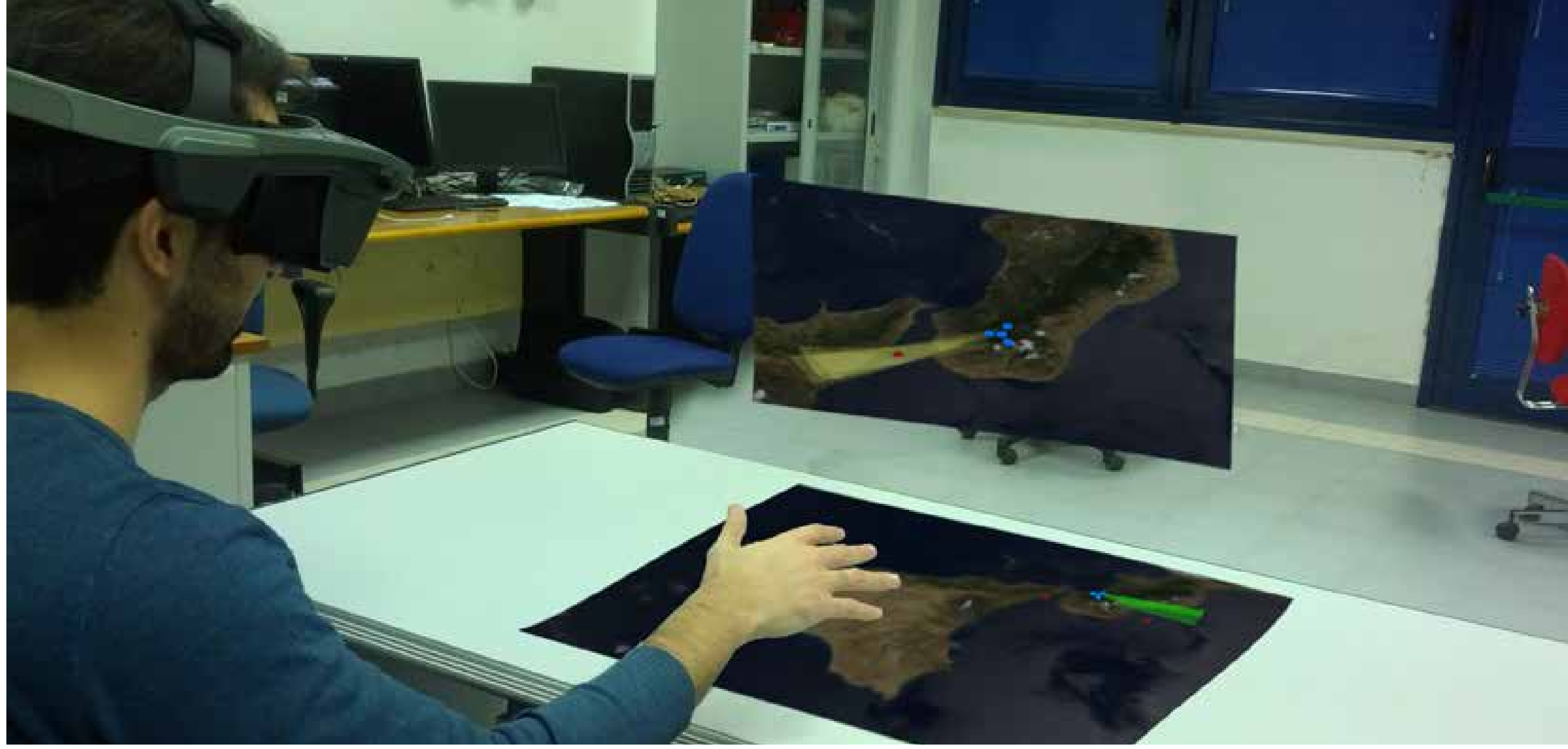


Figure 4 - Fanuc robot code generated at the end of the programming process



Augmented Reality
as added value of FEM results

Augmented reality provides a real-time interactive and immersive 3D experience (which can involve one or more of human senses) where real world objects are combined with additional computer-generated information and virtual objects. It allows delivering of contextually relevant information, which would not be directly available in the standard real world perception, to help the user in performing several tasks.
Augmented reality applications can exploit many different devices such as common personal computers, displays, input and tracking devices. Users can wear some visualization devices, such as head mounted displays or smart glasses, that offer a combined vision of real and virtual objects thanks to a small display placed in front of the eyes, but also it is possible to use the handheld displays, such as smartphone, PDAs and tablets, that can hold in the hands offering important advantages due to their portable and ubiquitous nature. In addition, mobile augmented reality applications can exploit their integrated cameras and sometimes GPS units to acquire data about real world scenes and show an augmented visualization on their display.
Besides entertainment, augmented reality has been adopted in several fields.
In minimally invasive surgery it can provide a doctor with a sort of "X-ray vision" of the internal organs on the patient body.
In education it allows to simulate physical phenomena that cannot be easily reproduced in a laboratory and to improve spatial abilities of students.
In military aircraft, it is used to superimpose vector graphics upon the pilot's view of the real world.
In cultural heritage, it shows additional information about a monument or a masterpiece people are visiting. Alternatively, it can also show how a monument should have appeared in a previous historical age.

Augmented reality could be useful in the visualization of results deriving from the application of the Finite Element Method (FEM). FEM is a powerful instrument to solve any problem that could be modelled as a set of partial differential equations, which are turned into a set of linear algebraic equations. Its application deals with many fields such as civil engineering, fluid dynamic, thermodynamic, electrostatic and electrodynamics.
The idea is to exploit the results provided by means of the numerical simulations performed within an advanced FEM environment allowing an augmented data visualization in order to permit the assessment of the multi-physical performances of a specific structure or component.

Lucio De Paolis, Valerio De Luca - Università Salento

For more information: Fabio Rossetti, EnginSoft
f.rossetti@enginsoft.it

INTERNATIONAL
CAE
CONFERENCE

OCTOBER

2016

STAY TUNED
FOR UPDATES!

WWW.CAECONFERENCE.COM