ENGIN SOFT

Newsletter

Simulation Based Engineering & Sciences

Year 13 n°2 Summer 2016

The optimization experience in the America's Cup Challenge

Increasing the **energy efficiency** thanks to CAE simulation in Immergas

Simulation experience in **air technology**

CAE modelling of twin Shaft **Concrete Mixer** MSO-RHYNO

Rocky DEM Helps PepsiCo **Cut Costs** with Virtual Product Testing

Forged Steel Valves for Energy Industry

Virtual ROPS and FOPS testing on agricultural tractors according to OECD Standard Code 4 and 10

FLASH

Once again, the picturesque and historically rich city of Trieste played a fitting host to the 2016 modeFRONTIER Users Meeting entitled "Speed of change". The seaport is a remarkable reminder of Trieste's diverse history and how it has embraced each movement of change throughout its history and now embodies an enchanting blend of bold architecture and natural beauty.

The speed of change is evident all around us, as we observe continual improvement in many areas of life. None more so than in the area of simulation where it is being rapidly employed to fuel the accelerated rate of change, thus producing better quality and more reliable products to customers.

Finmeccanica, itself has undergone a name evolution, now known as Leonardo Aircraft, to better reflect itself as an innovation player in the Aerospace, Defence and Security industry. On page 8 they discuss how they are implementing modeFRONTIER to the lesser known aspects of aviation.

On page 18, we uncover how multinational snack and beverage company Pepsico turn to simulation to cut cost with virtual product testing. Whereas on page 53, we look at the use of simulation as an effective countermeasure against natural disasters.

There are no signs to suggest that the speed of change will slow down. In the articles featured in this issue a variety of organizations demonstrate how embracing this accelerated evolution with simulation will allow us to achieve more, reaching the new heights of expectation for technology to improve our day to day lifestyle.

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°2 - Summer 2016

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

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

www.enginsoft.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

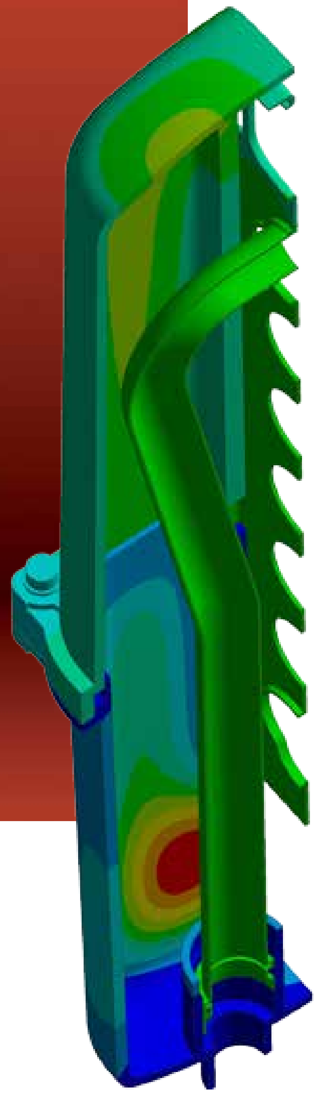
CONSORZIO TCN www.consorziotcn.it • www.improve.it
Cascade Technologies www.cascadetechnologies.com
Reactive Search www.reactive-search.com
SimNumerica www.simnumerica.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it

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

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



Increasing the energy efficiency thanks to CAE simulation

Interview with Eng. Luca Cavalli, Immergas, Advanced engineering team

Immergas has been the leading Italian company for condensing boilers since 2002 and before that traditional gas boilers. About 200,000 of the 350,000 boilers produced in 2015 are manufactured in Italy at the Brescello (Reggio Emilia) headquarters and most of the components are designed and made at our Italian plants, where over 600 employees design and manufacture 200 different models over 10 production lines.
Immergas also owns a second plant in Poprad, Slovakia, where it manufactures entry level appliances conceived specifically for emerging markets.
Similarly, the aim of the new start up being launched in Iran is that of producing boiler models suited to the Middle East markets.
Fifty years after its birth – on February 5, 1964 – the company started by Romano Amadei, Giuseppe Carra and Gianni Biacchi is consistently ranked among the leading European companies in the home HVAC industry and owns 100% of nine sales subsidiaries in Europe and one in China.

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

We have been using CAE technologies since late 1997. We first acquired the ANSYS software for FEA and the Fluent package for CFD simulation.
In 2004 we added the ANSYS CFX suite in order to increase the integration and the potential of the CAE package.
The first project developed with ANSYS Mechanical was an optimization of a pressure vessel made of stainless steel.

What was the main reason for introducing these technologies?

Since its inception the mission of Immergas is to win a leading position in the design and manufacturing of first-class products in terms of perceived quality, energy efficiency, safety and value for money in the extremely competitive market of the home HVAC.
I recall that in the late '90s we were well aware that globalization would bring unheard of opportunities and challenges such as larger untapped emerging markets, a more demanding and sophisticated customer base, increasing regulation and a shorter product life cycle. It was then easy to forecast increasing pressures on our R&D to deliver a wider range of appliances, with better performances, lower costs and reduced time to market.
Therefore I would say that the main reason for introducing the CAE was the necessity to invest in the engineering of new products in order to remain among the leaders and expand our customer base.

What kind of products are you using simulation for?

We use CAE methods to design and optimize a wide array of systems and components such as fuel mixers and controls, combustors, heat exchangers, exhaust piping, pressure vessels, filters and valves.
An emerging trend has been recently the downsizing of the boilers powered with fossil fuels and their integration into more complex systems that include renewable energy generators like PV panels, solar heaters or heat pumps.
Also in this case I believe that simulation can be of great help to understand the interaction among many devices and to ensure that each appliance performs in its optimal operating range and therefore to achieve the best balance in terms of performance and efficiency of the whole system.

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

As I have previously mentioned, our industrial sector undergoes increasingly competitive pressure.
Besides the usual requirements such as reliability, low cost and flexibility of the installation, the market demanded additional features such as energy efficiency, emissions, acoustic and connectivity to name a few.
Mathematical modeling can provide a large amount of data that can be used to understand the details of the physics associated to functional systems and components of the appliances for the HVAC.
Furthermore, by doing virtual simulations it is possible to explore a larger design space in a quicker way compared to the physical prototyping approach.

How does this affect your design process?

The design of an innovative appliance may require the use of many CAE tools such as advanced fluid dynamics, thermal, thermal-structural, mechanical, combustion, acoustics, DOE algorithms, etc.
Our internal advanced engineering team can source from a wide array of experimental data in order to validate the most effective simulation path.
Virtual modeling allows faster design iterations and only a few optimized solutions are tested in our laboratory.
Following the introduction of the CAE into R&D we have changed the design process remarkably from a trial and error method and physical prototyping loops to advanced collaborative engineering where the virtual simulation leads the process.

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

Today Immergas concentrates on intelligent installations that choose the more cost-effective energy source among condensing boilers, heat pumps, solar panels and photovoltaic energy in order to deliver heat or cool air with the least energy consumption.
New European and National legislative obligations and the need felt strongly by everybody to reduce pollution and the associated costs push towards more advanced technological solutions. Thermal solar and PV panels, heat pumps, hybrid systems and other sources of renewable energy are already well-established and in great demand.
We expect that these trends will grow stronger in the near future. I'm sure that Immergas, as we have been doing for almost 20 years, will keep investing in advanced engineering and will take advantage of the most updated CAE technologies such as Multi-physics integration, acoustics, advanced combustion and HPC.

What's the value that EnginSoft can give you?

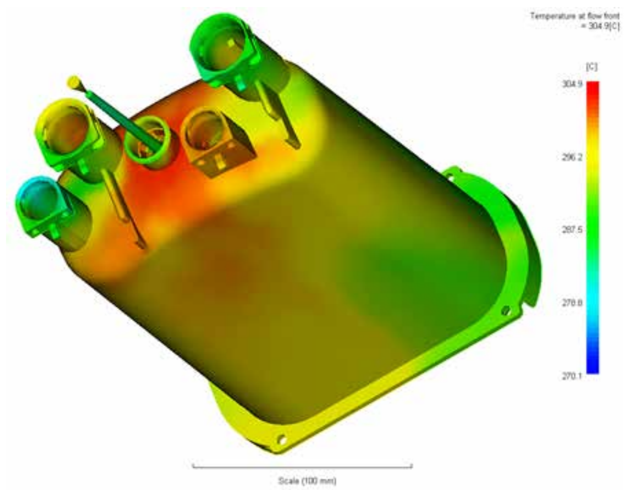
In our view EnginSoft's unique strength is the multidisciplinary approach that includes many areas of Computer Aided Engineering. Its expertise covers all types of analyses for both product and process, including all the key physics for our R&D such as advanced fluid dynamics, thermal, thermal-structural, mechanical, combustion, acoustics, FSI and multi-objective optimizations.
EnginSoft's collaboration with some of the most prestigious academic groups, leading industries and cutting edge CAE vendors insures it can transfer the best practices used in the most challenging engineering.
Besides the routine support provided on the simulations handled internally by Immergas, an other recognizable value of our collaboration with EnginSoft is the large amount of resources that can be quickly leveraged in order to speed the convergence of the projects toward their targets.
Lastly, the valuable partnership with EnginSoft is the continous push towards the education of human resources through a wide range of instruments such as training courses, training on the job, webinars and of course the annual CAE conference.

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

Looking ahead, we reckon that the home and commercial HVAC systems will be cleaner, more efficient and more connected.
Furthermore, the emergence of cutting-edge technologies such as IoT could enhance the demand of increasingly complex systems.
It is clear to us that the design challenges of the future, and the multiphysics phenomena involved, cannot be economically handled without an increasingly large portfolio of virtual prototyping tools.

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

Immergas develops a wide range of appliances in house or in collaboration with hundreds of suppliers and therefore it is hard to assess the CAE activities with a ROI figure without going through the challenges of each project.
I would rather talk about some of the costs of not doing simulations.
For instance, due to various constraints of our laboratory some high powered systems cannot be developed through physical prototypes.
Moreover, the performance demanded of our new products may easily go beyond the reach and the budget of a trial and error approach.
Last but not least the CAE adoption in Immergas has proven to enhance the skills and the commitment of our technical staff towards amazing innovations.



Simulation experience in aircraft technology

From Finmeccanica to Leonardo; a new brand that defines the evolution of the company and its transformation from a financial holding into an active integrated and innovative reality.
Selected for its evocative value, its name is inspired from Leonardo Da Vinci, universally acknowledged as a synonym of creativity and innovation. Leonardo is global company characterized by high technology and a key player in the Aerospace, Defense and Security industry worldwide Headquartered in Italy, Leonardo has over 47.000 employees, 37% of them working abroad in branch offices and industrial plants located in 15 countries, with a relevant industrial impact in Italy, United Kingdom, USA and Poland and with strategic collaborations in the most important and high-potential markets around the world.
Leonardo is organized in seven divisions: helicopters, aircrafts, aero structures, avionic and space systems, electronics for terrestrial and naval defense, defense systems, security and Information systems
In this way, Leonardo designs and produces a wide range of products, systems, services and integrated solutions to cover the customers' needs in terms of defense, protection and security in any other possible action: earth, sea, sky, space and cyberspace.

EnginSoft has interviewed the team of Environmental Control System (ECS) and Ice Protection Systems at aircraft manufacturer Leonardo's Aircraft division - Eng. Gaetano Mirra, Eng. Antonio Romano and Eng. Pierpaolo Borrelli

1. Can you give a quick snapshot of how modeFRONTIER is used to model some of the lesser known aspects of aviation?
Our group, ECS & Ice Protection of Finmeccanica Aircraft Division, is tasked with design and integration of environmental control systems, pneumatic systems, avionics and electronics equipment cooling systems, ice protection systems and oxygen systems, as well as thermal management design, including structural thermal management.
In recent commercial aircraft developments there has been a trend towards electronics systems integration characterized by higher heat densities and a more frequent use of composite primary structures. The overall airplane thermal efficiency is becoming increasingly important, since it affects several high level airplane requirements such as the fuel consumption, the direct operating costs and the noxious gas emissions. The passengers comfort requirements are a crucial aspect of current commercial airplane top level aircraft requirements. All these factors require robust airplane thermal management and thermal architecture design already at the preliminary design stages. In this context the improvement and optimization of the thermal architecture is regarded as one of the key success factors for future aircraft developments.
We have successfully used multidisciplinary optimization methods and tools, coupled with CFD, to optimize airplane conditioned air distribution network and air outlets in order to achieve high levels of passenger comfort, both in terms of airflow distribution in the passenger cabin and low noise, while increasing the pneumatic efficiency of the network.

Fig.1 -Team of Environmental Control System at Leonardo

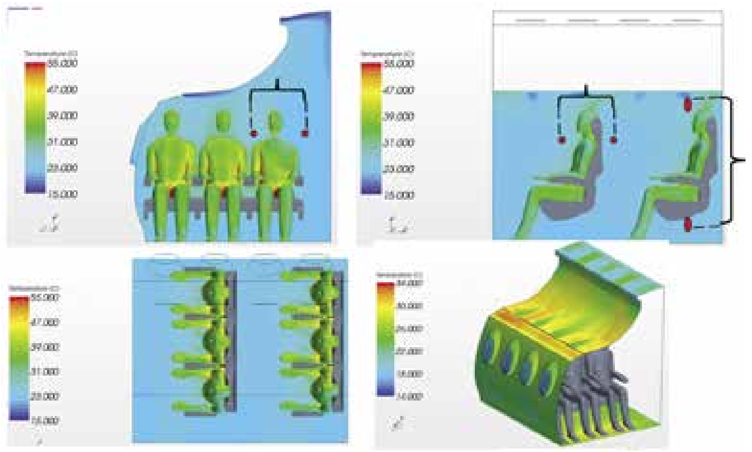


Fig.2 -Temperature pattern inside aircraft

2. All too often, when people think of aircraft simulation, heavy CFD analysis features. How does the simulation of these lesser known aspects differ?
Our recent experiences have shown the increasing importance of CFD analyses in many disciplines and technical areas where traditionally CFD was not used, such as airplane pneumatic systems and air conditioning design, avionics equipment cooling design, airplane thermal architecture design.
The complex phenomena that require CFD analyses in these disciplines have required the use of these analyses already at preliminary design stage, in order to rapidly mature our airplane architectures and increase the level of confidence on the robustness of our design before entering detailed design phases, where design changes are bound to have a high economical impact on the airplane project. Also, we are increasingly making use of CFD analyses in support of airplane certification activities, as a complement to testing and to reduce the testing phases, by helping us designing test set-ups, by reducing the number of test points etc.

3. What advantages does simulation bring when modeling these components?
The use of simulation (either CFD simulations or system-level 1D simulation) in all life cycle phases of the design process implies several advantages. On one hand, the use of simulation and multidisciplinary optimization in early design phases allow us to more precisely evaluate our design performances and margins, increases the robustness of the systems taking into account the design space constraints, more rapidly converge on the final design with confidence that the design won't have to be changed in later phases.
On the other hand the use of simulation in later stages of the design, including certification, allow us to dramatically reduce the development time and cost, for example by complementing the testing phases with simulations, as we said before.

4. How do you balance passenger comfort with optimizing the design of such components?
In one of our recent experiences we used the multidisciplinary optimization coupled with CFD to optimize the airplane conditioned air distribution network and air outlets. In such instances we have to balance the comfort of the passenger (which is one of the final goals of the design) with space constraints, imposed on the air conditioning distribution network by surrounding equipments and components, and with other constraints, such as the noise generated in the network and in the outlets. The optimization methods and tools as modeFRONTIER have helped us to achieve the best possible compromise between noise generation in the outlets, flow uniformity on the passenger and pneumatic efficiency of the air distribution network.

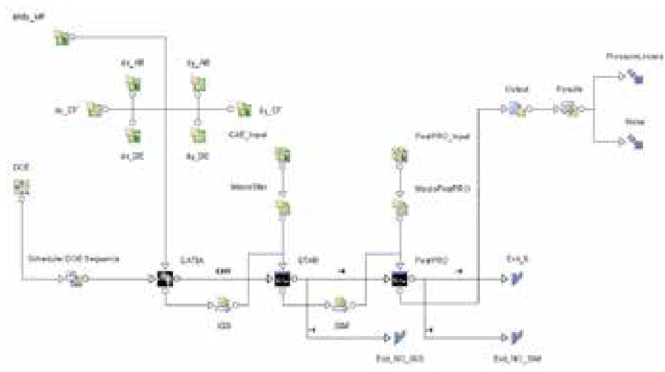


Fig.3 - modeFRONTIER model for shape and noise optimization of the outlets

5. And what other challenges do aspects present to the world of simulation?
Certainly the complexity of shapes of some of the components we design, the multi-disciplinarity of the design process and, in some cases, the sheer size of the physical phenomena are the main challenges we face, especially if you couple it with spatial constraints which define the design space.

6. Going forward, how will you address these challenges?
Some of these challenges we are addressing through super-computing. The use of HPC has made possible analyses which simply we could not have performed with the level of detail we would need, in an industrial program time-frame.
Certainly we expect in the near future to have coupling methods that ease the task of coupling different simulation tools, each of which may address one or more specific physical phenomena, to obtain a relatively easy-to-use multidisciplinary simulation framework.
On our side, it will become more and more important to engage the simulation tools vendors and developers, maybe through research projects, to "let them know" what are the challenges we face now or are likely to face in the future so that they may stream their developments also according to our present and future needs.

Faster than the wind: the optimization experience in the America’s Cup Challenge

1. New America’s Cup regulations: a design challenge

The 34th edition of the America’s Cup was a breakthrough event in the world of sailing, with traditional mono hulls giving way to the AC72 class foiling catamarans equipped with foils and wing sails. Since then, sailing and engineering teams have been dealing with a new set of challenges ranging from boat handling, tactics and, it goes without saying, the design of these new vessels and their subsystems.

From a design point of view, naval architects and engineers have been forced to rethink their way of working and open up to other design processes and methods, like in motor racing, which has already gone through a similar shift, where regulations tend to trigger a series of small incremental changes rather than radical one-off developments.

Moreover, the change from yacht to flying catamaran has revolutionized sailing philosophy, leading to constant changes in speed and boat response to conditions. This means that catamaran performance needs to be maximized by taking into consideration a whole new set of predictions and external factors. When the Luna Rossa Challenge Team started developing the concept for the catamarans in view of their campaign of the 35th America’s Cup, it opted to implement design process integration and automation routines. The limitations imposed by America’s Cup regulations served to highlight the need for simulations and multi-domain analysis - tools that proved crucial to developing and improving the new AC62 class boats.

> The automatic process was developed using modeFRONTIER. [...] This set up enabled us to use up to 400 cores for each design, significantly reducing the computational time from 10 hours to about 40 minutes
>
> Michele Stroligo
> CFD Analyst

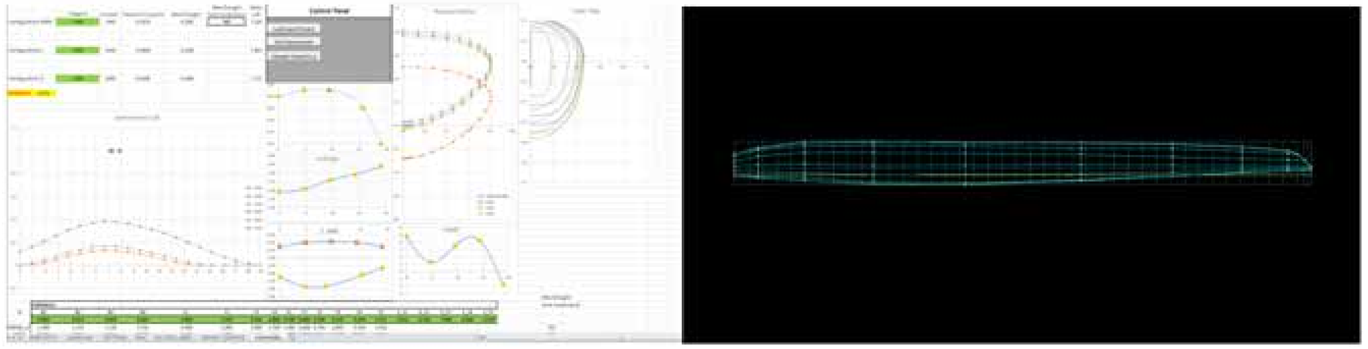


Figure 1 - Hull Parametrization

2. The sailing modes and the need for optimization

The new race regulations have brought about a multifaceted design process which requires taking into account different “sailing modes” and their respective physics in parallel. Even though the impact of the hull on overall performance at high wind speeds is practically negligible, its impact becomes significant at low-to-medium sailing speeds. Whereas in displacement sailing mode, the hull is fully immersed and more than 80% of the lift is due to the buoyancy of the hull, in skimming sailing mode, wind intensity makes the boat to start flying, resulting in a reduced effect of the buoyancy to 20% of the lift force. In foiling mode, at high wind speeds, the hull is completely out of the water and the catamaran sails on foils, reaching 30 knots upwind and 50 knots downwind.

Analysts therefore need to consider both hydrodynamic and aerodynamic drag when switching from one mode to another, meaning that the higher the number of different configurations in terms of hull, foils and wings considered as design alternatives, the higher the probability of enhancing the performance of each mode. Moreover, given that regulations prevent the actual sailing of 62-foot catamarans until around five months before the competition, most of the important early design decisions are necessarily based on data taken from simulations. The highly sophisticated design skills needed and the different disciplines involved in the design make performance prediction harder, leading to the conclusion that the use, coupling and automation of simulation tools in the design process are indispensable. Add to that the sheer number of variables, constraints and objectives involved and it becomes obvious that a trial and error approach is unfeasible. These considerations led the Luna Rossa Challenge Team to adopt modeFRONTIER as its automation and numerical optimization tool of choice, ensuring an integrated design approach from the earliest stages of the catamaran design process.

3. The Design Program
Hull shape optimization

As mentioned earlier, the hull is still a crucial element in the design of the boat. In the first stage of the design process the team decided to focus on the hydrodynamic analysis, considering the displacement and skimming modes. It is in pre-start phase when the hull shape affects performance the most as the boat accelerates from an almost static condition to reach peak speed and in some of the maneuvering conditions where the wind is not strong enough to make the boat fly. To optimize the hull shape taking into account the two sailing conditions, the team developed a hull shape generator to simulate the response for each variation and calculate the drag considering exclusively the shape.

Michele Stroligo, CFD Analyst at Luna Rossa Challenge, set up the logic flow with modeFRONTIER to drive the design investigation and optimization of the hull shape. He first prepared VBA macros in Excel to generate the set of control points and splines. These were then transferred to Maxsurf to create the surfaces and return a geometry file as output. CFD simulations were then computed with STAR CCM+, analyzing a single hull 3D geometry with a time-dependent simulation where the boat was free to sink, moving from the hydrostatic to the dynamic equilibrium.

“The automatic process was developed using modeFRONTIER, taking advantage of the Excel direct integration node, and two scripting nodes piloting the Maxsurf routine and the execution of CFD simulations on a remote cluster. This set up enabled us to use up to 400 cores for each design, significantly reducing the

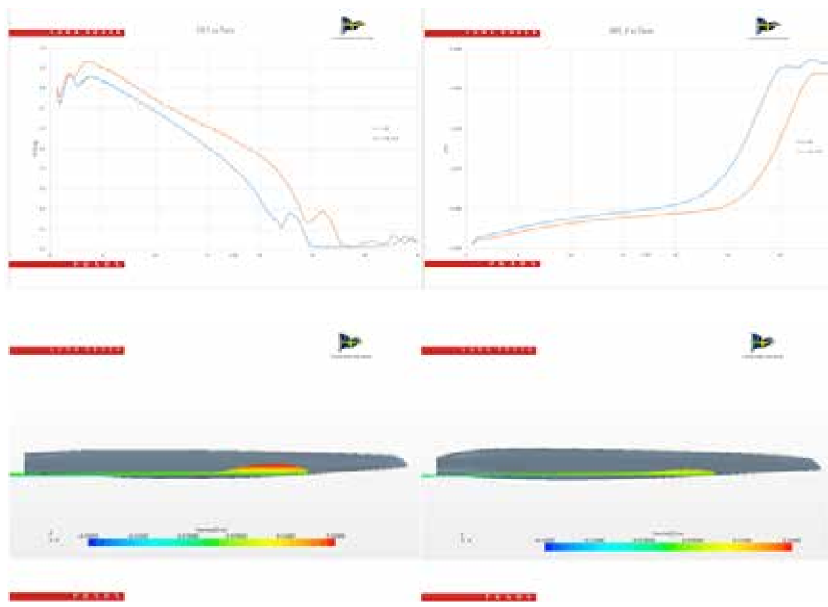


Figure 2 - CFD Acceleration test comparison between the baseline and optimized hull configuration

computational time from 10 hours to about 40 minutes" says Stroligo.

The results from the first design step showed a reduction of drag of the order of 2% in displacement mode and of 18% in skimming mode.

A single-objective process was used in the preliminary phase, where the cost function was weighted on each of the two computed conditions making this solution a compromise between the two scenarios. In the second step, the use of a multi-objective approach gave the advantage of making the solution independent from the user-defined weight, imposed previously. The geometries generated during this second optimization study ensured better results for the combined displacement and skimming conditions.

Moving forward, the team wanted to make sure that even during dynamic acceleration and take-off, the new candidates would bring about the same improvements when compared to the reference hull shape.

With this in mind, the team performed a series of acceleration tests using a mathematical model that simulated wing and sail loads and the related force that pulled the boat in order to determine the time needed to switch from skimming to foiling mode.

An appended hull configuration was used (hull, daggerboard, rudder and elevator) for these simulations. The angles and extensions of the appendages were the same for both cases. The comparison between a baseline hull and an optimized hull is shown in the chart below.

As highlighted in the image above, the optimized hull (right) confirmed its superiority also during accelerations and take-offs, enabling the catamaran to begin the foiling phase about 5 seconds earlier, giving an advantage in terms of speed, distance traveled and agility.

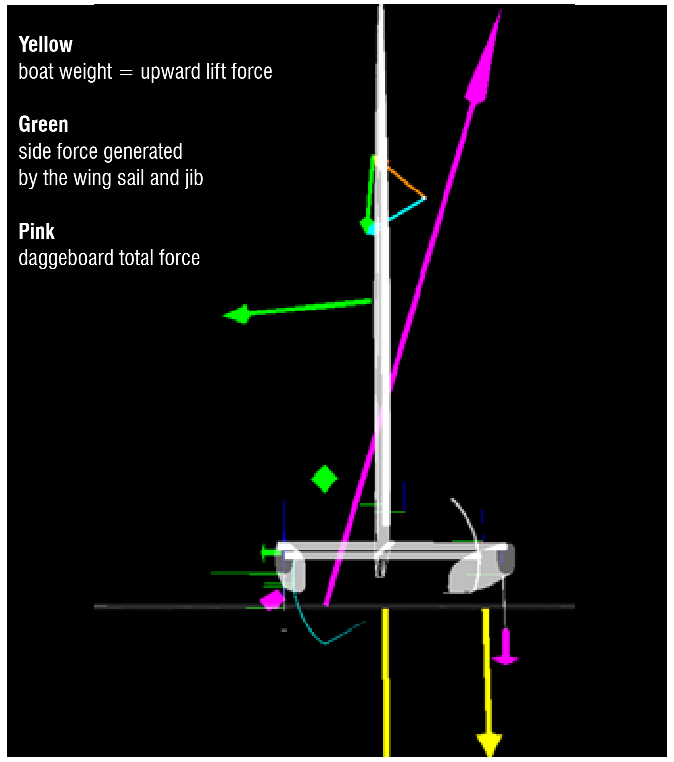


Figure 3 - Force scheme acting on the vessel

The exploration became fully automated, allowing for significant time savings
Giorgio Provinciali
Velocity Prediction Program (VPP) Leader

Foil optimization

The other major task of the design program at Luna Rossa Challenge was to maximize performance during in foiling mode. The use of daggerboards - or foils - enables boats to lift both hulls out of the water and "fly" in medium and high wind intensity. From a physical perspective, foils must ensure a sufficient upward lift force - approximately equal to the weight of the boat - as well as a high horizontal force to counteract the side force generated by the wing sail and jib. At the same time, the drag and roll moment had to be minimized.

To be complete, the analysis also needed to take into account constraints coming from rule specifications, structural behavior, cavitation limitations and stability criteria.

"At Luna Rossa Challenge, we managed to setup a workflow that helped us explore a very wide range of foil shapes in an attempt to identify the optimum shape for given targets (drag, heeling moment, VMG…) and subject to a number of constraints (rule compliance, structural, cavitation, stability….). In this way, the exploration became fully automated, resulting in significant time savings" says Giorgio Provinciali, Velocity Prediction Program (VPP) Leader, in charge of the foil design.

The optimization workflow for the foil was built by integrating a Rhino 3D/ Grasshopper model to generate the parametric 3D geometry; a CFD code (Panel code / Ranse) then evaluated the hydrodynamic performance.

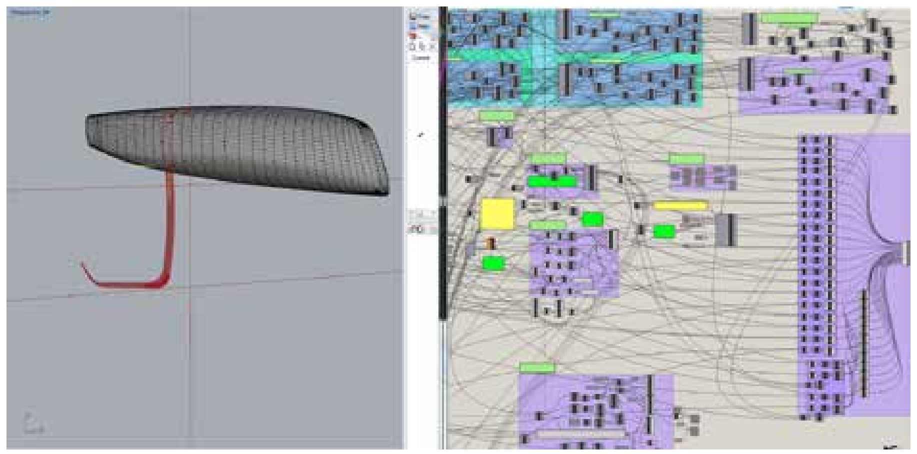


Figure 4 - Foil geometry and parametrization in Rhino and Grasshopper

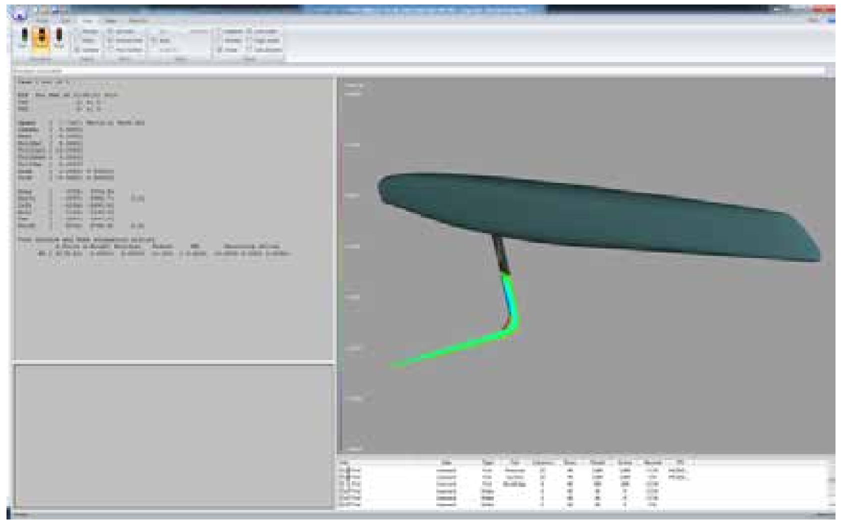


Figure 5 - Foil CFD simulation

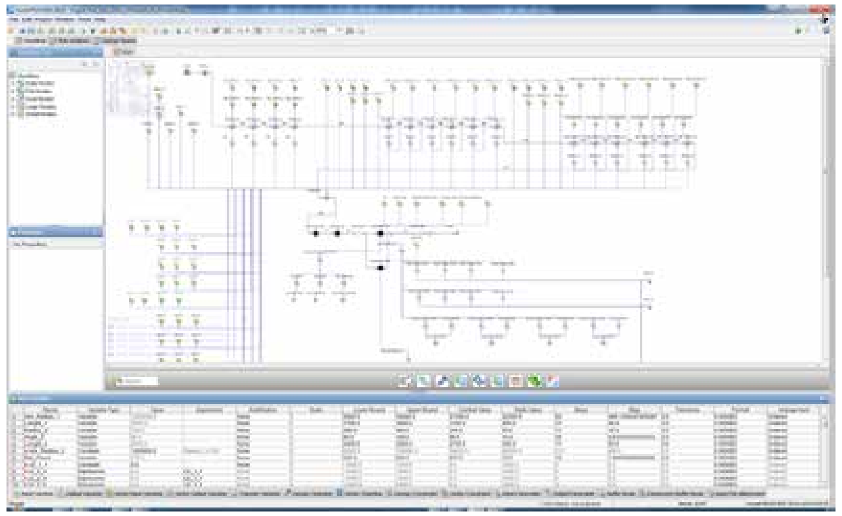


Figure 6 - modeFRONTIER foil optimization workflow

The geometry generation was driven by a script defining – among others - the following parameters:

- A spine curve defining the front view of the foil
- The leading edge shape
- Chord values along the span
- Airfoil thickness values along the span
- Airfoil camber values along the span
- Airfoil twist values along the span
- Airfoil sections basic shapes along the span

The file was read and run by a Grasshopper script within Rhino 3D and the updated .igs geometry file was then transferred to the CFD code selected for the simulation - either the in-house panel code (DasBoot) or Ranse (StarCCM+). When the panel code was used, leeway and rake capable of achieving a target lift and side force were sought for different speed values. Whereas with the Ranse code, the simulated values for leeway and rake were interpolated to find the target lift and side force at given values of speed. The optimization objectives were drag and roll moment minimization at different speeds determined by the upwind and downwind sailing configuration for a given wind condition. These conditions were estimated by weighting each wind condition with the expected wind distribution at the competition venue.

All inputs, geometrical variables, constraints and objectives were defined in the modeFRONTIER workflow. To successfully handle the highly constrained physical problem and efficiently explore the design space, the team opted for a combination of the ESTECO proprietary HYBRID and the NSGA II genetic algorithms. By taking advantage of the internal and automatic RSM computation of HYBRID, execution time was reduced even further. Despite the pervasive constraints, the algorithm was able to find feasible and efficient solutions and identify the Pareto front, balancing the optimal solutions for the two objective functions.

"The post-processing tools available in modeFRONTIER gave us a good grasp of the most important parameters impacting the objectives and their correlation. Even more so, these advanced tools clearly highlighted the design trends, putting us in the right direction for more detailed investigation."

4. modeFRONTIER benefits and conclusions

The America's Cup regatta showcases the best sailing and engineering teams in the world who push design and vessel performance to the limits in their aim to win the coveted competition. Relying on design and simulation tools has become unavoidable; however, choosing the technology that serves as a true enabler of a designer's ingenuity is still an invaluable source of advantage against other teams.

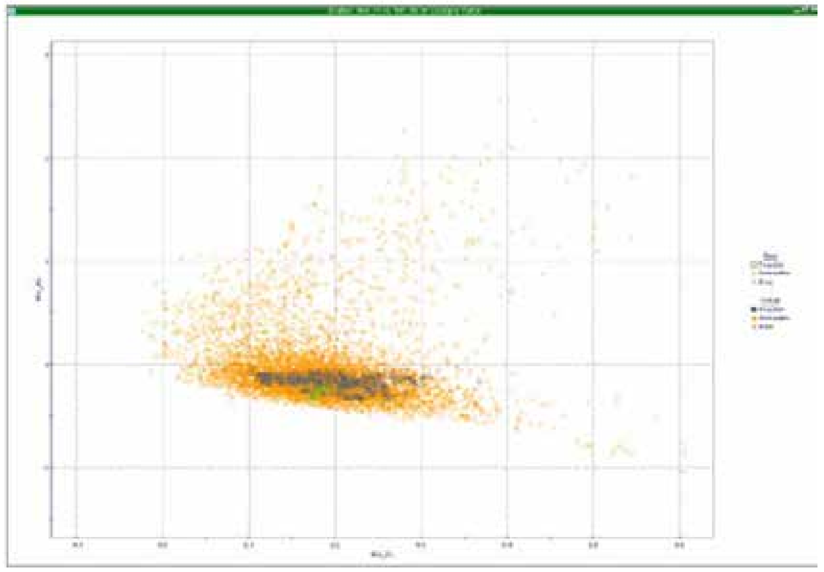


Figure 7 - Pareto Front for drag minimization and roll moment

As highlighted in the case studies, modeFRONTIER gave Luna Rossa specialists four key advantages: the automation of the design processes, the seamless integration of the software chain, the effective exploration capabilities of its proprietary algorithms and – boosting the efficiency of the whole simulation process - the flexible handling of distributed computing resources. By integrating and automating the multiple tools, engineering team were able to let the complex, multi-disciplinary simulation workflows run autonomously and simultaneously consider several physical aspects while having more time to focus on design analysis, post-processing of results and in-depth decision making.

The intelligent design space exploration and optimization capabilities of the algorithms combined with the efficiency of using a distributed computation set-up helped reduce the development time and quickly delivered prototypes to be tested by the sailing team. By running parallel simulations on a network of computers using the modeFRONTIER Grid Tool, designers found better solutions with a reduced number of iterations made by the robust algorithms. Further steps of the design program at Luna Rossa aim to include the other disciplines (structures and aerodynamics) as well as other modeling approaches (VPP simulation, race modeling program, wing sail optimization and boat handling) in the process.
Provinciali concludes that "working on the Velocity Prediction Program (VPP) and race modeling within the foil design optimization workflow would allow us to optimize boat performance by also considering the race track and the wind conditions expected at the AC venue." Stroligo points out that in this perspective, the efficient management of the parallel execution of simulations, automated with modeFRONTIER, opens up to a broader analysis which can take into account a larger number of objectives including those concerning maneuver and handling requirements.

Giulio Cassio, ESTECO

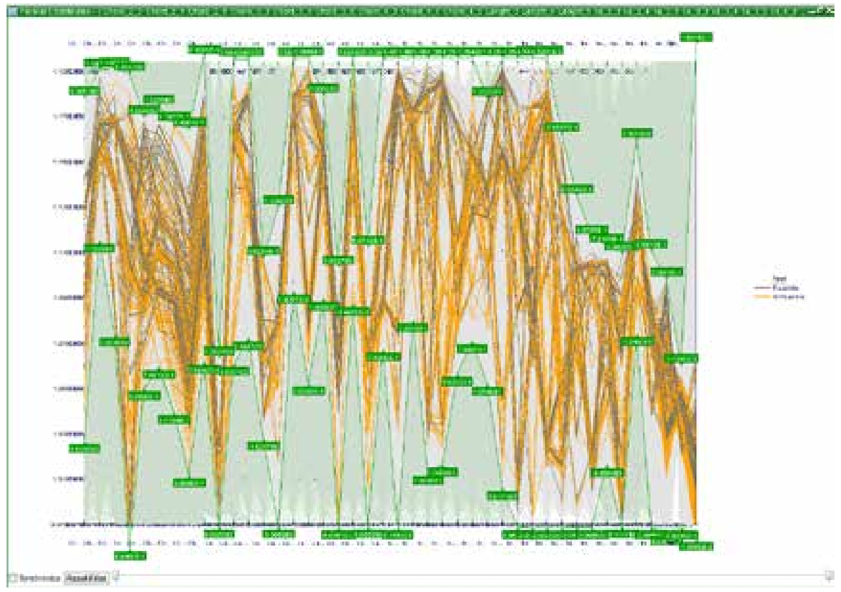


Figure 8 - modeFRONTIER Parallel Chart for feasible design analysis

Figure 9 - Sensitivity analysis of parameters impacting drag and roll moment

For more information on modeFRONTIER Industrial applications:
Francesco Franchini, EnginSoft
f.franchini@enginsoft.it

The Designers
Giorgio Provinciali | VPP Leader
Member of Luna Rossa's Design Team from 2001 to 2007 and, as consultant, he contributed to the victory of BMW Oracle Racing in the 2010 America's Cup. In the 34th America's Cup he was in charge of the appendages design and the performance prediction program for team Luna Rossa.

Michele Stroligo | CFD Analyst
Naval architect and marine engineer specialized in CFD and hydrodynamic studies applied to yacht design.
In 2010 Michele joined Oracle Team USA where he carried out hydrodynamic CFD simulations for hulls, appendages and rudders.
In 2014 Michele joined Luna Rossa Challenge Team for the 35th America's Cup.

CAE Modelling of twin Shaft Concrete Mixer MSO-RHYNO

For concrete production cement, sand, small stones or gravel and water are commingled to a homogeneous self-hardening mass. The mixing is the most critical phase of concrete production, quality of the final product, attributes are depend on the mixing performance.
In this article we describe the research project carried out by SIMEM in collaboration with EnginSoft, on the twin shaft mixer MSO-RHYNO for a design improvement with CAE approach.
SIMEM is a world-leading company on designing, manufacturing machines and plants for concrete production, materials processing and the environment protection. Over the last 50 years SIMEM develop four Business Units:

- CONCRETE BATCHING PLANTS: to create advancing mixing technologies;
- WATER TREATMENT PLANTS: to develop solutions for technical water, water treatment and water recovery;
- WASTE TREATMENT PLANTS: to develop treatment for both solid and liquid waste;
- MINING PLANTS: to offer equipment and solutions for mining industry.

SIMEM is the reference company for concrete batching plants. Technology, innovation, reliability, quality and post-sale services are factors that distinguish SIMEM and that make it unique in the sector. Following this policy SIMEM has successfully reached its targets in the supply of worldwide plants solutions for big projects such as the EuroTunnel, Petronas Towers in Malaysia, the Olympic Stadium in Sidney, the Airport of Hong-Kong and many others.

Twin shaft mixer MSO-RYNO technology

The batch mixer with double horizontal axis is the best solution for concrete production in terms of quality, cost and reliability of guaranteeing an absolutely homogeneous mixture in a less time than any other of the mixing system.,The twin shaft mixer MSO-RHYNO rapidly produces mixtures with consistently high levels of homogeneity. It is designed to ensure reliability and reduced service and maintenance costs.

Figure 1 - Twin shaft mixer MSO-RHYNO

The objective of the project was the improvement of mixing efficiency for the actual paddles configuration with the development of a new layout. Some geometrical dimensions were developed as parameter in the computational domain, in order to be modified for additional simulations, in particular the junction angle between the paddles and the shaft.
A baseline configuration and the impact of two geometrical changes (rotation and shape of the paddles) have been compared in terms of mixing efficiency and mechanical stress on the rotating elements.

The CFD study

The CFD simulation involved a multiphase, transient, numerical model to characterize a liquid phase inside the mixer and the overlying gaseous phase.

Concrete is a mixture of solid and liquid components and within this project it was modeled as liquid phase, considering equivalent physic-chemical properties. In particular concrete is a non-Newtonian fluid that can be described with the Bingham plastic low. This type of fluids have a linear shear stress/shear strain relationship (Equation 1) and require a finite yield stress before they begin to flow.

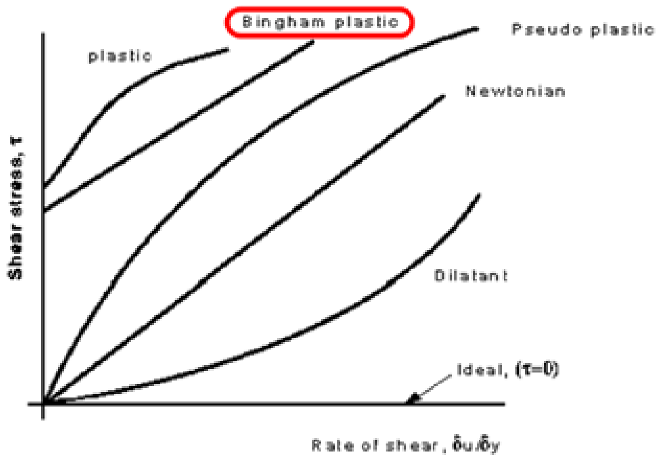


Figure 2 - Shear stress function of shear strain

$$\tau = A + B\frac{\delta u}{\delta y}$$

Equation 1 - shear stress/shear strain relationship for Bingham plastic materials

The A and B coefficients of Equation 1 were calculated starting from experimental data of the slump test considering one particular concrete.

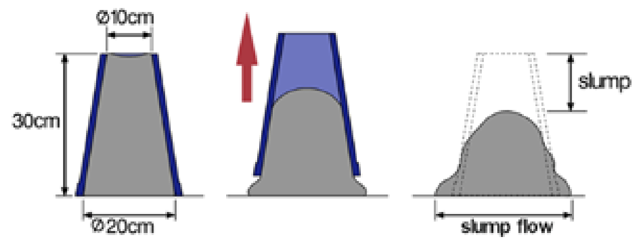


Figure 3 - Slump test

The initial conditions consider the system at rest and the rotary elements (shafts and paddles) have been put in explicit rotation with a law of motion function of time.
The analysis allowed to understand the shape and the evolution of interface between liquid phase and gaseous phase and the velocity field; the presence of recirculation zones and stagnation area was used as an index of uneven mixing. At the same time it was possible to evaluate the stress of the rotating parts, in terms of forces and moments. After verification of the baseline configuration, geometrical changes have been introduced to improve the flow mixing with an acceptable increase in stress.
Case 2 introduced an additional paddle at the end of each shaft, in order improve the orthogonal mixing that exchanges material between the left and the right side of the tank.
The third model (Case 3) is an additional evolution and introduces a new profile for the central paddles in order to increase the working surface; the objective is an improvement of the axial flow trying to limit the required torque.

The quantification of the mixing efficiency was evaluated with an injection of inert tracers (virtual ink) within the liquid phase and some results are reported in Figure 5 and Figure 6. Tracer 1 (red) and Tracer 2 (green) are injected at two different location; the goal of SIMEM was the traces should flow axially around a shaft and then flow towards the other side of the tank in a recirculation motion. The baseline configuration indicates that the shafts are correctly working, but the final distribution of the two tracers is significantly different. Since the symmetrical configuration of the system, the mixing efficiency of the two tracers depend on the position of the paddles at the injection instant, so the same distribution of Tracer 1 will be obtained even for Tracer 2 changing the injection instant and vice versa. Case 3 is characterized by a better evidence of the recirculation motion and at the same time a reduction of the dependency of the position and instant of injection has been achieved; the computation of the standard deviation for the tracers confirms an improvement of the mixing for the new paddles.

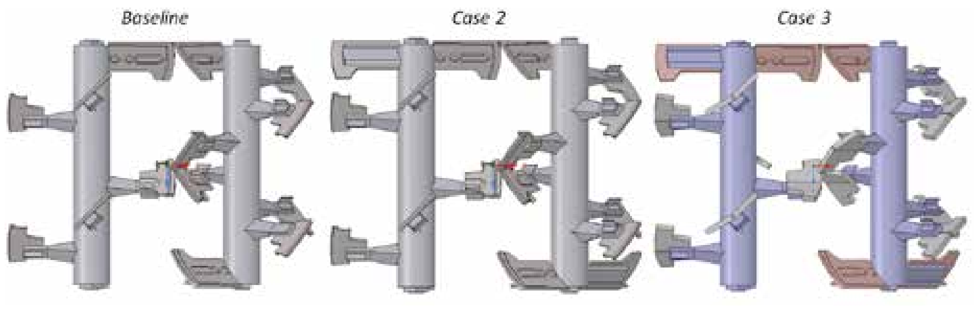


Figure 4 - Geometric configurations

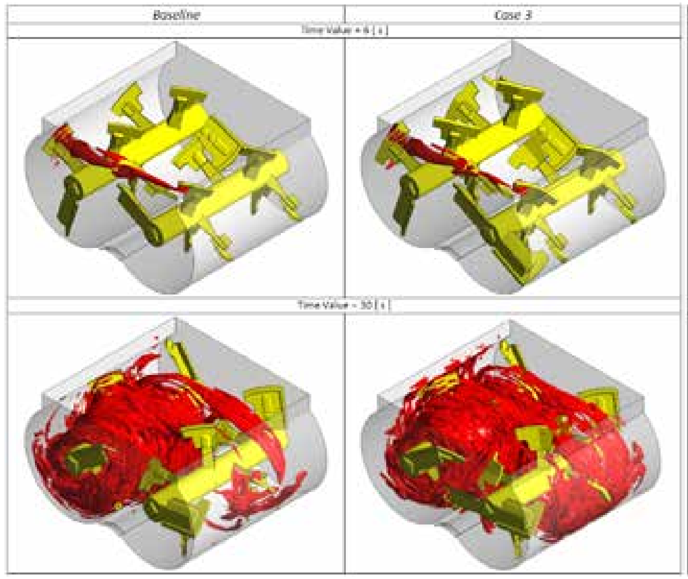


Figure 5 - Tracer 1 mixing

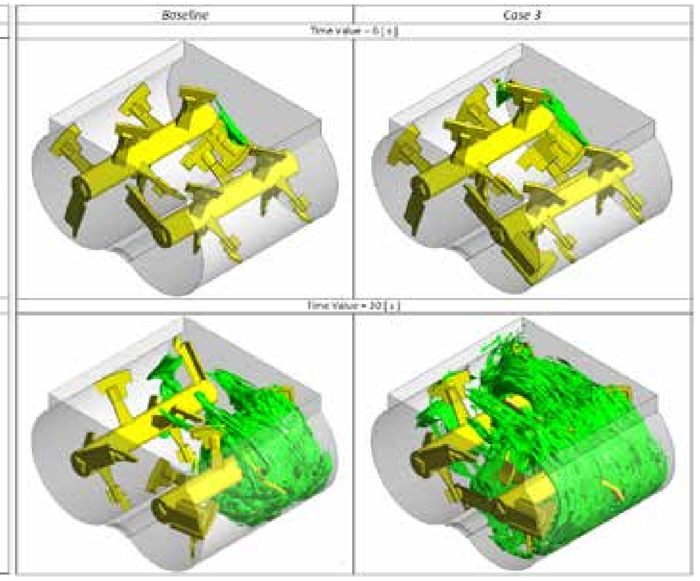


Figure 6 - Tracer 2 mixing

$$\% = \frac{Volume\ of\ air - entrained}{Total\ concrete\ volume} \cdot 100$$

	Baseline	Case 2	Case 3
3s	5.26%	6.78%	8.01%
10s	14.28%	15.79%	17.25%
20s	19.83%	21.69%	23.65%
30s	21.20%	24.20%	26.06%

Table 1 - Air content

During the mixing process the paddles continuously breaks up the air into bubbles and drag them into the mix and the air content in the concrete was used as another parameter to evaluate the mixing quality. The geometrical changes introduced in Case 2 and Case 3 increase the air- entrained.

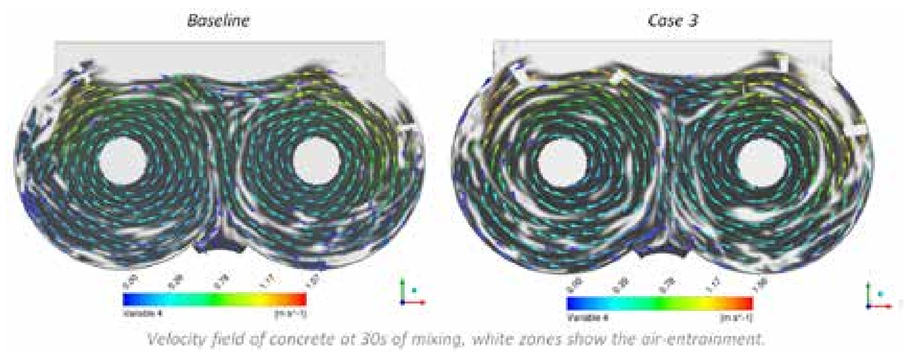


Figure 7 - Slump test

The results obtained by the fluid dynamic simulations, in particular the pressure fields on blades and shafts, have been used as input data, for the structural simulations.

Through analysis with the finite element method (FEM), the stresses and deformations of the rotating parts of the mixer were evaluated paying attention to the junctions between components (bolting between paddles and arms and bolting between arms and shafts). The results on the three design configurations analyzed do not show significant structural problems.

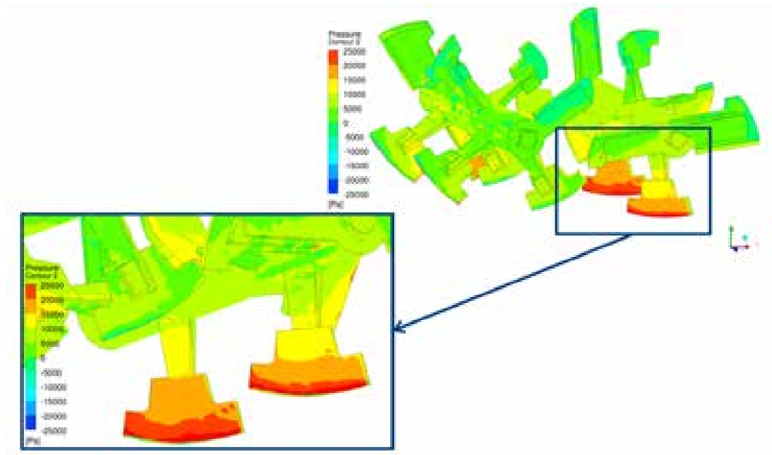


Figure 8 - Pressure field on blades and shafts.

Conclusions

A innovative paddles configuration for the twin shaft mixer MSO-RHYNO has been designed by SIMEM in collaboration with EnginSoft.
The CAE approach allowed to compare the fluid dynamic behaviour, pressure fields, torque required and mechanical stress of different configurations and the new design indicates an increment of the mixing process. Therefore, the selected configuration is a reliable solution that permit to confidently realize the first prototype.

For more information:
Claudia Quadri – EnginSoft
c.quadri@enginsoft.it

Rocky DEM Helps PepsiCo Cut Costs with Virtual Product Testing

Simulating realistic concave shapes enables the company to solve several materials handling challenges

More and more companies rely upon simulation tools to help develop high-quality products and reduce costs. PepsiCo, Inc., one of largest multinational food, snack, and beverage corporations in the world, is using these simulation technologies to analyze and improve products and processes. "*We have leveraged Rocky DEM to help our engineers understand and resolve a number of material handling challenges*," says Chris Koh, Director and Global R&D Fellow at PepsiCo. "*When the challenge is too complex for theoretical or experimental approaches, we turn to simulation to virtually perform experiments. Virtually testing the feasibility of the new product or process allows us to significantly streamline our innovation workstream.*"

Custom Concave Particle Shapes

DEM (Discrete Element Method) is a useful method of simulating how particle-based interactions perform at the bulk solids level of a production line. A primary limitation of legacy DEM solvers is the lack of realistic particle shapes. These DEM packages would limit usefulness in PepsiCo's application, but the company solved this problem by using Rocky DEM, the only DEM package capable of simulating truly non-round particles and concave shapes.

"*One of the advantages of using Rocky DEM is the ability to model more realistic curved/concave shapes, an important feature of many of our products*," says Koh. "*Using simulation models with simplified (i.e. flat) shapes can lead to omission of important physical interactions between uniquely-shaped materials, such as snack chips, and the processing equipment.*" To more accurately represent concave materials like snack chips, PepsiCo creates a unique geometry in a 3D CAD package and then imports that custom shape into the Rocky DEM software.

Virtual Tests

One of the major ways Rocky DEM helps PepsiCo improve productivity and reduce costs is by allowing the company to perform virtual experiments on their product lines. "*When new processes are developed, much of the product produced cannot be sold and must be scrapped*," Koh explains. "*By having the capability to do a large number of the developmental experiments virtually, the amount of waste produced can be minimized*."

The company uses Rocky DEM to evaluate all kinds of equipment issues related to speed, vibration, and material distribution, and to correct problems like material clogging and breakage. In this way, Rocky simulations helped PepsiCo optimize the tumbling process used to apply seasonings to their snack chips, as well as improve upon their packaging process.

	Real-World Concave Shape	Physical Representation in Rocky DEM simulations
Snack Chip		
Extruded Corn Snack		

Fig. 1 - PepsiCo uses custom concave particle shapes to represent their various food products

Return on Investment

To be able to perform virtual experiments is very important to any company handling bulk materials, because by testing a large of number of possibilities, they can find the best solution to the problem, identify improvements, increase productivity, and reduce costs. PepsiCo found all these benefits by using Rocky DEM. "To run physical trials in our pilot or production facilities can be quite expensive, due to labor and material costs, especially when a large number of simulations are needed to complete a Design of Experiment (DOE)," Koh explains. "Having the ability to do many of these experiments virtually can save both money and time. Depending upon the situation, we have realized up to a 50% cost reduction when leveraging Rocky DEM simulations."

Challenge

Standard DEM particle shapes did not accurately capture the behavior of unique materials, like PepsiCo's snack chips, on the production line. Alternative solutions, such as glued spheres, presented an unfeasible node count for even test-batch-sized simulations.

Solution

By developing an accurate polyhedral contact detection model for curved shape concave particles that can be solved using Graphics Processing Solver (GPUs). Rocky DEM enabled PepsiCo to create unique shapes that not only accurately represented the real-life material but also made it feasible to use it in large scale process simulations.

Benefits

Rocky DEM simulations enabled PepsiCo to improve the accuracy of their factory equipment set-up for both their new and modified product lines. The simulations enabled them to reduce the number of adjustments required and helped them identify any potential custom modifications ahead of time. Between avoiding costly modifications, and reducing product waste and time during production testing periods, Rocky DEM has helped PepsiCo realize up to a 50% cost reduction thus far.

Testimonial

At PepsiCo, when new processes are developed, much of the product produced cannot be sold and must be scrapped. By having the capability to do a large number of the developmental experiments virtually, the amount of waste produced can be minimized. To run physical trials in our pilot or production facilities can be quite expensive, due to labor and material costs, especially when large numbers of runs are needed to complete a DOE. Having the ability to do many of these experiments virtually can save both money and time. Depending on the situation, we have realized up to a 50% cost reduction when leveraging Rocky DEM simulations.

About Pepsico

PepsiCo began in 1965 with the merger of Pepsi-Cola and Frito-Lay. Today, it's a global food and beverage leader, with a portfolio that includes 22 brands.

About Rocky DEM

Rocky DEM, Inc. is the developer of Rocky, 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. www.rocky-dem.com

Rocky DEM is distributed by EnginSoft. For more information:
Massimo Tomasi, EnginSoft - m.tomasi@enginsoft.it

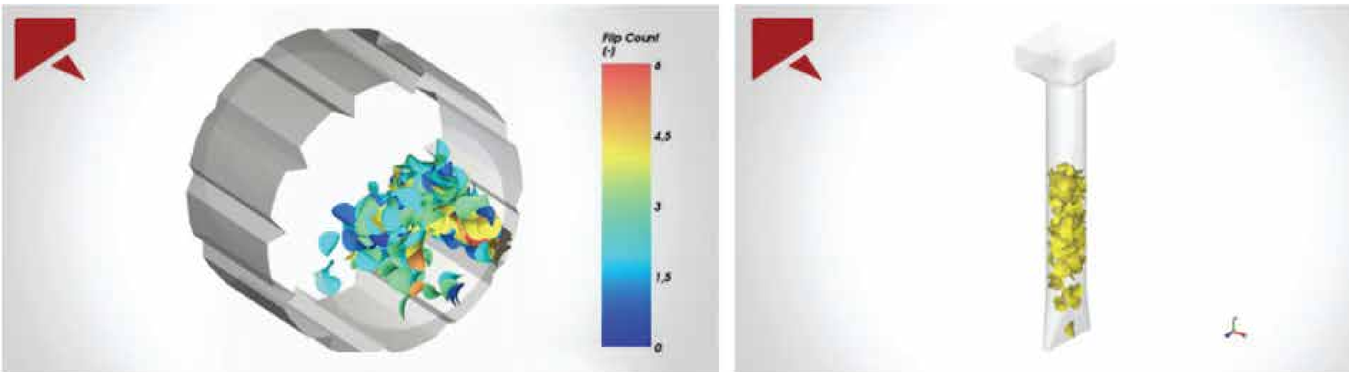


Fig. 2 - Rocky DEM simulations of snack chips at different stages of production, including in a seasoning drum (left) and being filled into a bag (right)

Forged Steel Valves for Energy Industry

We have been collaborating with EnginSoft for many years now, to optimize our products using numerical simulation. The achieved results over the years have led us to improvement in terms of products performance, also reducing the re-design needs and modifications on on-going projects. For all these reasons and to keep our leadership in the ever changing sector of energy, we have decide to invest in ANSYS technology, which is, according to us, the most complete and reliable multi-physic tool.

Eng. Cristiano Sana
Product Engineering Manager, OMB Valves Spa

Valve Specialists™

The OMB Valves group, headquartered in Cenate Sotto, Bergamo, Italy, is a diversified manufacturer of valves for the energy industries. Founded by Mr. Roberto Brevi in 1973, OMB is a family owned and operated group which has become a globally recognised manufacturer of forged steel valves for the oil&gas industry. The group operates 3 plants in Italy (OMB, Fluicon and Calobri), one in Singapore and one in Stafford, Texas. OMB has direct presence in UK, Japan, Korea, China and Canada with its own subsiadiaries and a distribution network which covers all the major oil producing countries. The OMB group has knowledge and experience in design and manufacture of valves for the hydrocarbon and petrochemical processing plants, oil and gas off-shore and on-shore production and transportation and the energy industry.

ANSYS in design

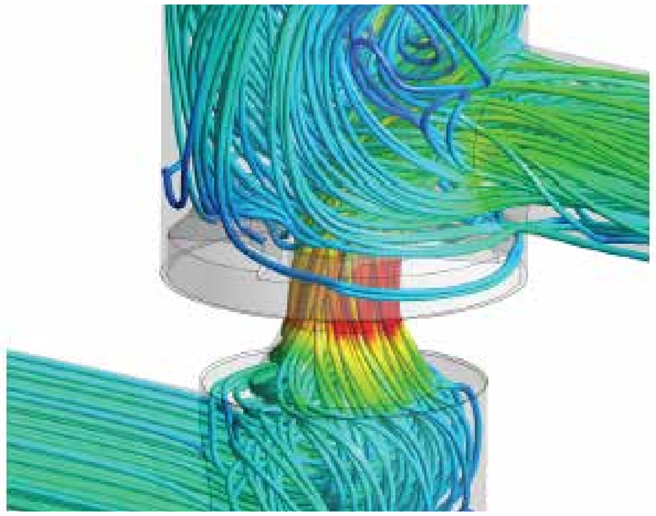
The FEM analysis is a fundamental requirement when dealing with Oil&Gas valves, both as far as the design and the resistance verification phase are concerned. For OMB Valves, FEM simulations have allowed to verify gate valves, globe valves and ball valves at various load conditions and to analyze their resistance before starting the experimental tests. The FEM simulation of the load conditions and the related post-processing have also permitted to verify that the different valve components (cover, valve body, gate, ball, seats and bolts) could bear the necessary loads and if any change should be performed during the design phase instead of during the experiments.

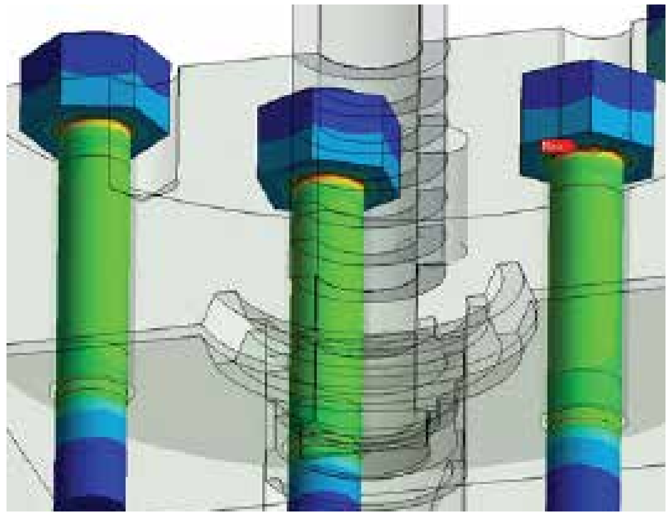
With ANSYS Workbench and ANSYS Simulation, it has been very easy to define the materials of the valve components, the contacts (between the different bodies) and the boundary conditions (as for instance the bolts preload, the fluid pressure, the piping loads, etc.), as well as the thermal loads to the valves (as for the thermal gradient between external and internal temperature, it is particularly critical in the case of cryogenic valves). Also the interaction between mechanical and thermal analysis could be easily defined, thus allowing us to evaluate the simultaneous effect of mechanical and thermal loads as requested by the regulations.

Other simulations have been performed on valves, in particular CFD analysis by means of the ANSYS CFX code, that have enabled us to calculate the flow coefficient cv and any pressure drop of the valve, so to estimate the impact of the valve on the whole system flow.

It's important to underline that ANSYS Workbench allows both FEM and CFD analyses within a single environment, for a really automated simulation phase.

The main advantage of ANSYS is its reliability, which makes ANSYS the leader in the Oil&Gas/Energy sector as far as simulation is concerned. ANSYS always keeps on improving its software in order to meet any new customer's request. ANSYS provides the ability to work out both the modelling and the post-processing in accordance to the ASME regulation, which are the most common ones in this sector worldwide. This element has permitted OMB valves to face the global market, without adhering to any other regulation or peculiar customers' specification.





Virtual ROPS and FOPS testing on agricultural tractors according to OECD Standard Code 4 and 10

1 Introduction

The main goal of the Roll Over Protection Structure (ROPS) and Falling Object Protective Structure (FOPS) is to provide protection to the operator in case of roll-over accident and falling objects. Such passive safety features are commonly found in agricultural and forestry tractors and are conceived to protect the operator from a serious injury or even death in case of an unexpected accident. Agricultural accidents may be caused by improper maneuvers, hill falls, road accidents and in such cases the protective systems must be able to absorb the impact energy without endangering the driver.

In an effort to improve the operator's safety in agricultural and forestry tractors, the Organization for Economic Co-operation and Development (OECD) has established worldwide some standards to harmonize the protective equipment testing and therefore facilitate international trading. Since its foundation in 1961, many countries have joined and agreed on a wide range of standards beyond agricultural.

In this case, the OECD Code 4 and 10 have been numerical studied in relation to tractor ROPS and FOPS. The use of Code 4 sets up the requirements in terms of energy or force for the longitudinal, lateral and vertical directions of the cabin structure, while Code 10 implies a series of drop tests to test the upper head protection against falling objects. Besides structural resistance, both codes define a clearance zone where the driver should be seated, which has not to be entered by any part of the structure or impacting object at any time.

There are several reasons why LS-DYNA has demonstrated to be suitable tool to investigate the ROPS and FOPS performance in the early stage of the design phase:

- Robust contact algorithms.
- MPP scalability (Massively Parallel Processing).
- Available material models.
- Full-restart feature.

The current article will show the main aspects regarding the ROPS and FOPS modeling with LS-DYNA. The correlation between experimental and numerical results will be shown for the ROPS study as there was no experimental data available the FOPS test during the investigation. The tractor has since been approved for both ROPS and FOPS and it is currently available in the market under the Landini brand.

2 Brief tractor description

The protective structure (ROPS) is mainly made of a reinforced tubular welded steel frame which is joined to the tractor chassis by means of the platform. The platform is the lower part of the protective structure which is fixed to the tractor by four supports. Silent blocks are mounted on the front and rear supports in order to provide cushioning and therefore comfort to the driver.

Fig.1 - CAD model of the tractor showing the protective structure FOPS

the kinetic energy of the system remained below the total internal energy a couple of orders of magnitude. Hence, minimizing the inertial effects.

It is pointed out that the required energy is the product of the pusher's displacement by the necessary force to deform the cabin. Each load step implies a loading and unloading phase. The key aspect about ROPS testing is that all loading steps are linked, that is, the deformation of cabin at the first loading step will influence the following ones and so forth. Thus, only after the unloading phase, the location of the pusher for the next load step can be determined.

In other words, the position of all the pushers cannot be determined in advance since it depends on the deformation history of the cabin. For instance, the user cannot predict the necessary displacement to fulfill the energy requirement in the first load step, therefore, such displacement has to be tracked down and once achieved, re-run the simulation with the unloading phase (simply moving the pusher backwards).

Fig.2 - ROPS sequence – longitudinal, rear, side and front loads, respectively

The protective structure (FOPS) is an assembly mainly made of plastic materials which is designed to provide overhead protection to the driver.

3 Test description

3.1 Code 4 - ROPS

According to OECD Code 4, the ROPS tests shall be conducted in the following sequence:

1. Longitudinal loading
2. Rear crushing
3. Side loading
4. Front crushing

For further details about the pushers (geometry, location), clearance zone etc. the reader is referred to the Code 4 reference.

Longitudinal and side loadings have to fulfill the requirement of energy absorption, while rear and front crushing have to sustain the prescribed loads. In any of those cases, the clearance zone (central green box in Fig.2 has not to be entered by any part of the structure to pass the test. The magnitude of the required energy as well as crushing force depends upon the reference mass of the tractor, in this case, 4400 Kg. The rate of load application shall be such that it can be considered as static. This was numerically done in the simulation by carrying out a quasi-static analysis. That is, using the explicit dynamics solver, the pusher's speed was selected so that

Sequenza	TEST	VALORI
1	Longitudinal loading	E_{L1} = 1,4·M= 6.16 [kJ]
2	Rear crushing	F = 20·M= 88 [kN]
3	Side loading	E_{IS} = 1,75·M= 7.7 [kJ]
4	Front crushing	F = 20·M= 88 [kN]

Fig.3 - Loading sequence for Code 4. M (reference mass) = 4400 Kg

The ability to output restart files by LS-DYNA is of great value since can save lots of CPU time. In particular, the full-restart allows the inclusion of new parts such as the pushers. The only keyword that the user needs to set up is the *STRESS_INITIALIZATION to initialize the old parts in the restart phase.

In the Fig.3 the loading sequence is summarized along the energy and force requirements.

3.2 Code 10 – FOPS

The OECD Code 10 states that the drop test object shall be a spherical objected dropped from a height sufficient to develop 1365 J. The drop object shall be made of solid steel or ductile

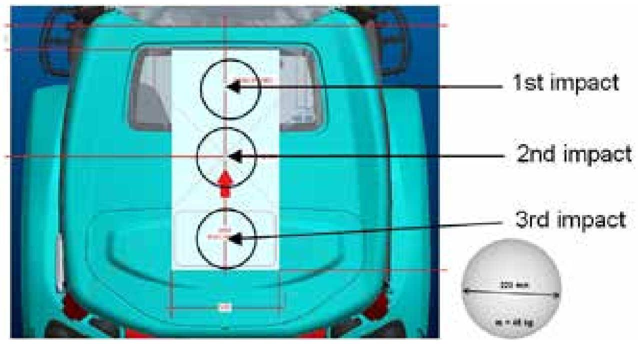


Fig.4 - FOPS impact sequence

iron sphere with a typical mass of 45 Kg and a diameter between 200 and 250 mm (220 mm employed in the current study). Three different impacts in sequence were chosen for the current investigation as shown in Fig.4.

4 FE Modeling

Two different FE models were created.
A first model was built up for the ROPS testing and a second one including the overhead assembly for the FOPS testing.

4.1 ROPS structure (cabin)

The cabin CAD geometry was accurately meshed with 2D shell elements since most of the components were tubular steel frames and thin metal sheets.

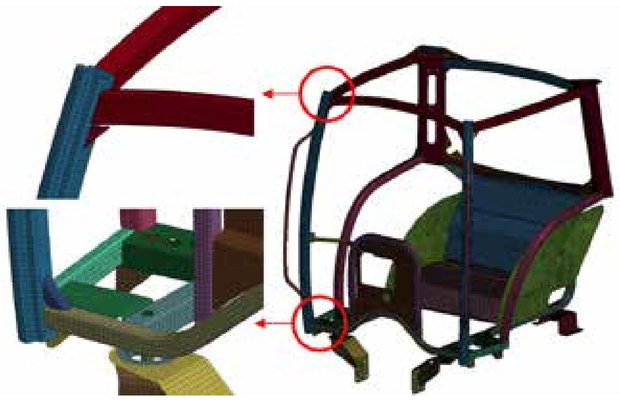


Fig.5 - Meshed main protective structure for ROPS and FOPS test

Average mesh size of 10 mm, corresponded to a good trade-off between accuracy and computational time. In fact, the average mesh size was reduced from 15 mm down to 5 mm to study the mesh influence on the obtained results. It was recognized that the 10 mm mesh size provided meaningful convergence.

The total number of elements corresponded to 80207 shell elements. Fully integrated formulation (ELFORM=16) was employed for the study.

Sheet metal pieces were connected (welded) using the *CONSTRAINED_NODAL_RIGID_BODY option in LS-DYNA. Therefore, welds were modeled as a non-breakable connections between parts. In fact, this approach was very useful at the early design stage to identity potential critical areas. In such cases, more detailed methods using continuum elements were used to evaluate weld failure. Nodal rigid body spiders were also used to model bolted connections.

*MAT_24 (elastoplastic) with experimental tensile true stress-strain curves was used to characterize the steels in the cabin. In particular, mainly three types of steel were employed: S-235, S-275 and S-355 (UNI EN 10025).

The easy to use *CONTACT_AUTOMATIC_SINGLE_SURFACE was used for all the cabin components. This contact takes into account potential self-contacts as well as contact between components (including shell thickness). From the user point of view, the definition of such contact only requires a set of *PARTs in the model so it can be immediately defined without the need to individually search for potential interacting parts.
The tractor was fixed to the ground by means of four supports.

4.2 Silent Blocks

The cabin is suspended by means of the rubber silent blocks, which are components for absorbing and dampening vibrations in order to increase the driving comfort. Two are located at the front and two at the rear.

The behaviour of the silent block was modeled with beam elements (ELFORM=6). The radial and longitudinal stiffness were set up according to experimental results. To do that, *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM was employed. Such material model allows for the definition of an arbitrary translational force curve along the local axis of the beams. Hence, it is very useful to define different axial as well as radial behaviour, including bottoming out, where the rubber material cannot further absorb energy and becomes stiff. This effect can easily be taken into account by means of a steep increase in the force vs displacement behaviour.

4.3 Pushers

In a ROPS test, the pushers are the components that transmit forces to the cabin in order to achieve certain amount of energy (force vs displacement) or simply, specific crushing force value.

The load to the ROPS structure needs to be uniformly applied by means of a stiff beam (normal to the direction of the load). Such beams are bound in order to prevent lateral displacement. Experimentally, as the load is applied (typically by means of a hydraulic system), force and displacements are recorded. Numerically, the force was the result of a prescribed motion to the pushers.

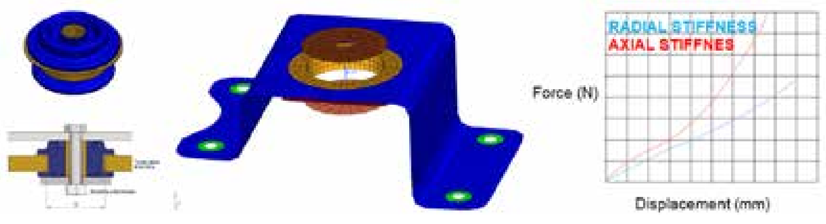


Fig.6 - Example of rear cabin support with the silent block modeling

Specifically, the pushers were modeled as a rigid components and a specific *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE was assigned between such component and the rest of the cabin. The universal type of joint between the hydraulic piston and the stiff beam was directly modeled using the constraint rigid body motion (CON1 and CON 2) included in the *MAT_RIGID (*MAT_20 card).

4.4 FOPS structure (roof)

The FOPS structure is an overhead assembly which is conceived to transfer The FOPS structure is an overhead assembly which is conceived to transfer the impact loadings due to potential falling objects to the cabin structure.
The model consisted of 338K shell elements varying from 2.5 to 6 mm depending upon the component. Full integrated formulation

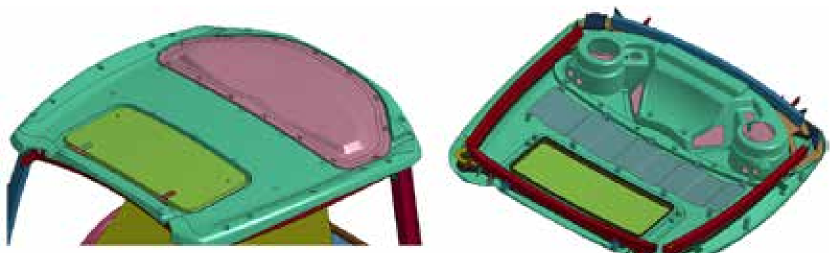


Fig.7 - Overview of FOPS protective structure

(ELFORM=16) was used. In this case, the mesh size was reduced to 5 mm to better capture plastic strain gradients on the components. The bolted connections between components were modeled with rigid body spiders (*CONSTRAINED_NODAL_RIGID_BODY).

Most of the materials encompassing the overhead assembly are plastics with the exception of the central reinforcement plate which is made of steel S-235. The plastic material was characterized by *MAT_24 (elastoplastic) where the tensile stress-strain behavior was added as an input curve. No failure was implemented in the model, so plastic strains along with engineering judgement were used to identify critical areas.

5 Results and Discussion

5.1 ROPS

In the following paragraphs (5.1.1 - 5.1.4) a comparison will be shown between the numerical and experimental approval results. It is pointed out that such experimental ROPS phase were only done after the numerical results met the code 04 requirements.

As a matter of fact, numerical forces as a result of the pusher's action were recorded with time and compared to the experimental results for the longitudinal and side loading. The prescribed energy was simply obtained by integrating the force vs displacement curve. Regarding the crushing tests, the compression force was applied at the corresponding location but with no displacement recorded. The likelihood of failure was evaluated by looking at the accumulated equivalent plastic strains during the test.

5.1.1 Longitudinal loading

As described in the Fig.3, the main goal of this test was to achieve 6.16 KJ of absorbed energy by applying a longitudinal load and most importantly, guarantee the clearance zone.
The overall behavior of the ROPS structure was in accordance to experimental results as depicted in the Fig.8. In addition to this, the force vs displacement curves were in agreement as shown in Fig.9. The stiffness of the cabin is well predicted in the first part of the test (see Fig.9) where the main tubular structure is in elastic regime. Nonetheless, once the stresses on the tubular components start exceeding the yield strength, the slope begins to decrease significantly in comparison to the experimental test. This difference may be mainly attributed to the material model as well as the cold forming history of the components which has not been taken into account. It is pointed out that no reverse engineering was done to tune the material model since the characterization was done prior to testing. Hence, further improvement of the results can be expected by numerical tuning of the steel materials but this was not the objective of the current study.
None of the plastic deformations found in the ROPS were considered to be critical (Fig.10). Concerning the clearance zone, none of the structural components entered into the volume.

Fig.8 - Numerical ROPS prediction vs real test for longitudinal loading @ 6.16 KJ

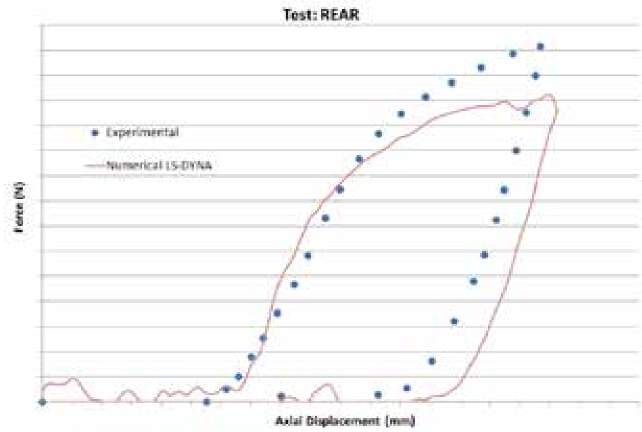


Fig.9 - Experimental vs numerical force for rear longitudinal loading (ROPS)

The critical state, which corresponds to the maximum longitudinal displacement at 6.16 KJ did not penetrate the clearance zone.

5.1.2 Rear crushing

The aim of this test was to apply a compressive load of 88 KN on the rear side of the ROPS structure.

Fig.10- Longitudinal loading. Grey: elastic region areas. Colored: main plastic strain areas

The ROPS structure was well designed as it could withstand the load 88 KN without compromising the clearance zone. On the other side, the new plastic strains introduced on the structure were limited and therefore not critical.

5.1.3 Side loading

According to the standards, after the rear test, it was sequentially applied the side loading to achieve 7.7 KJ of energy absorbed.
It is noticed that once a test is successfully passed, the pushers are positioned and the test is started. To take into account the history of the ROPS structure (plastic strains, stresses, and updated geometry) and speed up the engineering process, the full restart feature was sequentially used as well. Hence, there was no need to re-run the previous load steps to keep on with the ROPS study.
The ROPS structure showed good agreement in terms of deformations as well as loading response (force vs displacement). In this case, the numerical behavior of the ROPS structure seemed to be stiffer compared to the experimental test (Fig.13). As a matter of fact, the numerical curve is mainly above the experimental one, meaning that the energy time calculated by integrating the curve will be higher as well. Despite of this fact, the results were satisfactory.

Fig.11 - Numerical vs experimental rear crushing test

Fig.12 - Numerical vs experimental side loading test

New plastic strains were not critical based on engineering judgment and the clearance zone was preserved.

5.1.4 Front crushing

The final test of the ROPS sequence is the front crushing. As for rear crushing, the force applied to the front side of the cabin was 88 KN. No particular weak points were found in the structure and the clearance zone therefore guaranteed.
After such test, the ROPS structure numerically met the Code 4 requirements and therefore was ready for the approval test. In agreement with numerical findings, the experimental ROPS results did not show particular failure areas.

5.2 FOPS

In this phase, 3 sequential sphere impacts on the FOPS structure was numerically investigated. On the one hand, it was checked

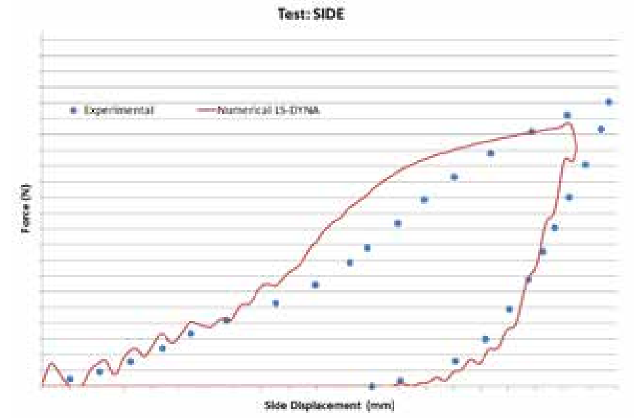


Fig.13 - Experimental vs numerical force for side loading (ROPS)

the maximum deflection of the overhead protection did not interfere with the clearance zone. One the other hand, maximum plastic strains were analyzed to assess the likelihood of failure since the material failure was not implemented in the model.

5.2.1 First Impact

For this impact, high plastic strain values were found at the pinned connection of the top hatch. At first glance such plastic strains may not seem critical but possible differences in mechanical properties (scatter) due to manufacturing reasons, may cause local failure. Simulations showed that the clearance zone is guaranteed provided that no failure occurs at the opening / closure mechanism. For such reason, although not shown in the current study, the mechanism was reinforced prior to approval testing.

5.2.2 Second Impact

The second impact guaranteed the clearance zone. Limited plastic strains were located at brackets of the metal sheet reinforcement. Such reinforcement was important to contain the maximum deflection caused by the steel ball. No critical areas were found.

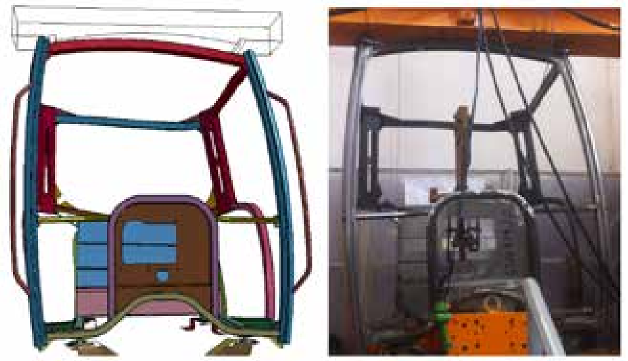


Fig.14 - Numerical vs experimental front crushing test

Fig.15 - Front crushing. Grey: elastic region areas. Colored: main plastic strain areas

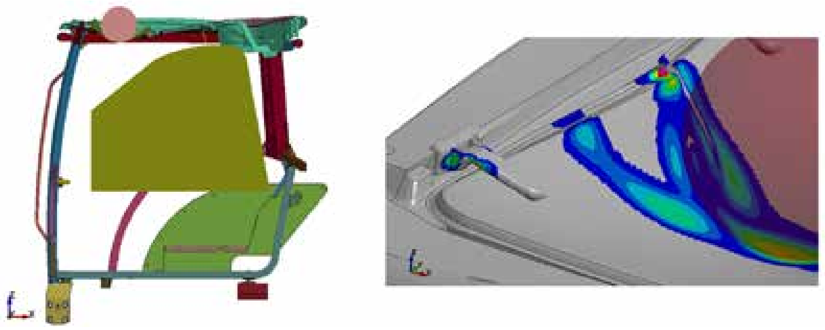


Fig.16 - Overview of FOPS first impact and main plastic strains on the structure, respectively

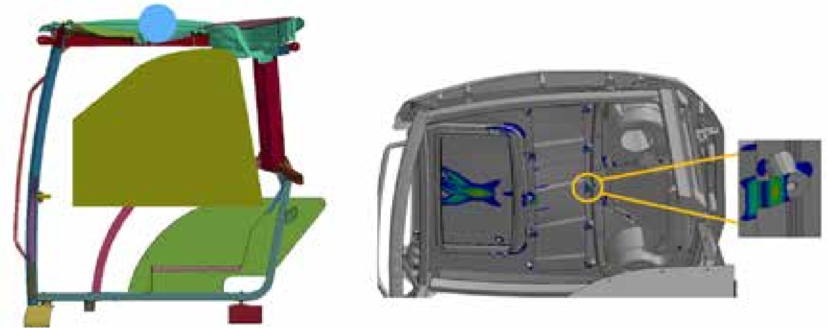


Fig.17 - Overview of FOPS second impact and main plastic strains on the structure, respectively

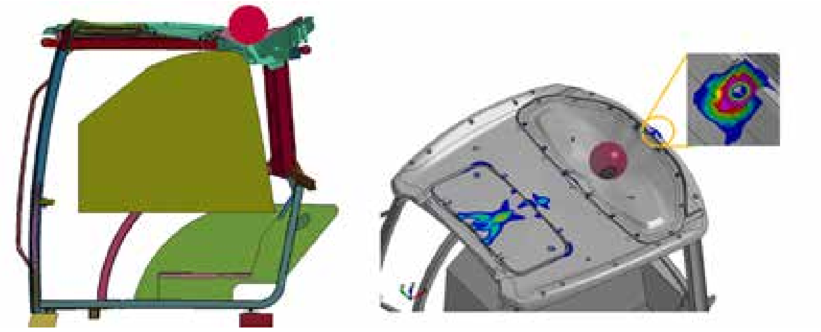


Fig.18 - Overview of FOPS third impact and main plastic strains on the structure, respectively

5.2.3 Third impact

Some critical areas in terms of plastic strain were found at top lid bolted connections meaning that local failure may occur. Moreover, despite of the fact that the clearance zone was guaranteed, the distance between the overhead assembly and the clearance zone (at the maximum deflection point) was not enough to face with confidence the experimental test. Hence, further improvements not shown in the current study were introduced to conservatively handle possible misalignments with the real test. After analyzing the first and third impact and implementing the modifications, the experimental FOPS approval test was done and passed.

6 Summary of tests

6.1 ROPS

All ROPS steps were successfully passed.

6.2 FOPS

All FOPS steps were successfully passed. However, some critical areas were found and the final overhead protection system was reinforced. The final version of the overhead assembly improved the results showed in the current study.

7 Conclusion

LS-DYNA was proved to be a very useful tool to predict the ROPS and FOPS behavior in the early stage design phase. In fact, ROPS experimental testing confirmed numerical findings in terms of loading response and stressed areas. Likewise, FOPS study give valuable insights into overhead performance and as result critical areas were reinforced.
It is important to notice that the user took advantage of the full-restart feature along MPP scalability (Massively Parallel Processing) to save CPU time and speed up the engineering process. In addition to this, the models were very robust (despite of large deformations, nonlinear material behavior and contacts) giving rise to consistent results.
The added value of simulation was demonstrated that the post processing results were used to give a cutting edge advantage over traditional design methods in order to assess performance. As a matter of fact, critical points were identified and modifications were done quickly (in FE ambient, without needing to build the CAD model) to improve results. By doing so, the behavior of the ROPS and FOPS structure under complex loading scenarios was understood and therefore the experimental approval tests were faced with confidence.
In the end, the tractor was approved for ROPS and FOPS standards and now can be found in the market.

D. Hailoua Blanco, C. Martin, A. Ortalda - EnginSoft
P. Venturelli - Argo Tractors S.p.a

Development of new shaped Ring Rolling process through FORGE NxT Numerical Simulation

Felb Srl has been established since 2006 from a demand of the market for medium weight forgings up to 10 tons. Reference market sector refers to medium size forgings (maximum length of 6 meters, maximum diameter of 2500 mm and maximum width of 2000 mm) and to rings and bushes(up to a maximum diameter of 4000 mm and a maximum height of 870 mm). The special equipment allows the production of shaped forgings according to drawings, bars of various sectional sizes, discs, shafts, rollers, molds forms, bushes and flanges. All manufactured pieces can be submitted to mechanical machining and heat treatment processes. Furthermore they can be tested by official authorities in the internal homologated test laboratory. The orders are usually based on few items, hence an high flexibility is mandatory to satisfy the customers' requests.

The requirements

FELB received an order from a new customer concerning the supply of about two hundred A105 steel shaped rings (Fig.1). Approx. dimensions of the machined part are 900 mm (diameter) and 500 mm (height).
The part could be produced by the traditional process composed by ring rolling and subsequent machining. The size of the request (one order of magnitude bigger than usual) led the management to evaluate the possibility to roll a shaped ring, in order to optimize material usage and to provide a final part with better in-service performances. Due to its limited experience on the shaped rolling process, the company evaluated FEM code FORGE NxT, developed by Transvalor S.A., and involving EnginSoft, Italy distributor and FORGE user for 20 years, in order to speed-up the training and to obtain the best results in very short period.

Figure 1 – Flanged body ordered by the costumer

The approach

Fabio Fioletti, material engineer and technical manager at FELB with extensive experience on design and simulation of forging processes with FORGE, developed the preliminary numerical analysis of the ring rolling process, in collaboration with the "Metal Forming Team" of EnginSoft. The aim of these analysis was the evaluation of the technical feasibility, particularly to obtain the upper flange starting from different preformed rings. Based on the company know-how on ring rolling process, the studies and simulations of several lamination curves (Fig.2) and geometries on king roll allowed the validation of the right feasibility by using the company equipment.
On specific terms, the required maximum torque on king roll and axial force on mandrel were verified, as reported in Fig.3.
These preliminary evaluations were completed in less than one week and allowed the order concerning the equipment supply (mandrel and king roll) to be issued. At the same time, simulation activities were focused on the evaluation of the best forging sequence to obtain the optimal preform, in order to reduce the required material. From this point of view, several modifications on flange height, both internal and external diameters have been evaluated. All the solutions have been subsequently validated by numerical simulations of ring rolling processes.
The equipment arrived within a few days after the completion of all the FEM activities and the production started immediately, with results very similar to the expected one (Fig. 5). The developed simulations allowed for a very fast fine-tuning phase, highlighting the causes of some filling defects on the flanges and suggesting the best actions to be performed in order to complete the profile on the real process. FELB immediately started the production of the ordered batch, respecting the agreed delivery terms.

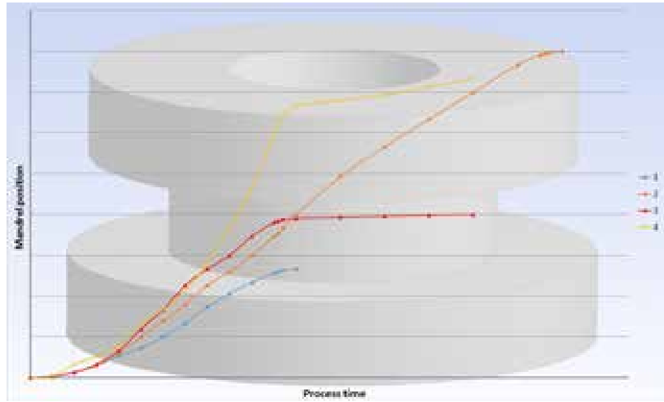


Figure 2 – Lamination curves evaluated on FORGE

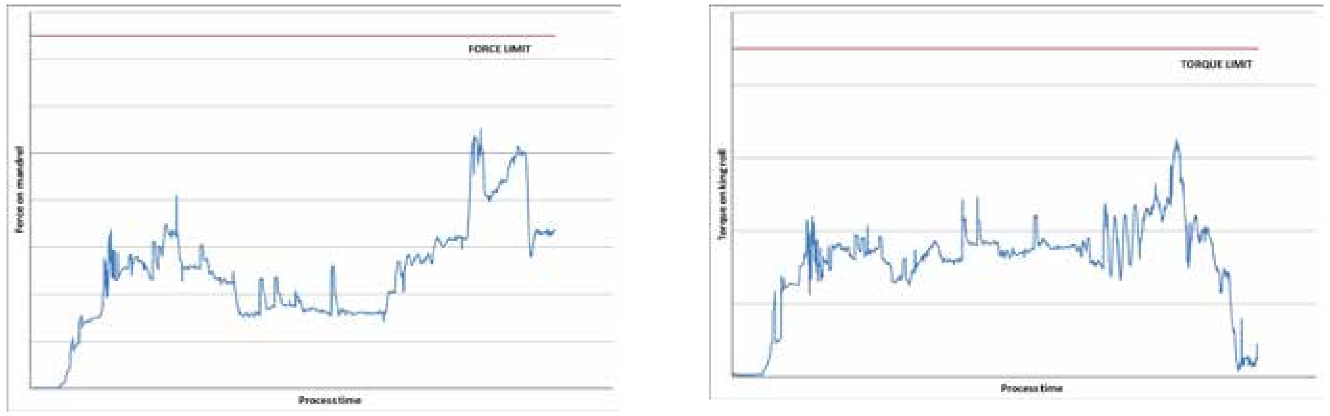


Figure 3 – Axial force on mandrel (left) and torque on king roll (right)

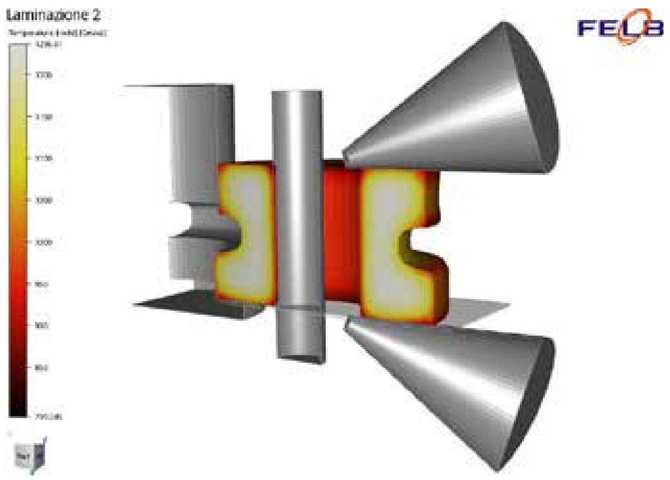


Figure 4 – Numerical simulation of shaped ring rolling process

Obtained results from Forge Simulation

The numerical simulations allowed to evaluate the technical feasibility of the part by forging and shaped ring rolling processes, besides the sensible differences on the part quality. Traditional process involved a massive machining which led to high material waste. Furthermore, machining trimmed the material fibres, leading to lower capacity of the ring to withstand to in-service loads. The innovative developed process leads to a minimum oversize and a better fibers distribution (Fig.6), more compacted and with no trims, for higher overall performances.
Verification on the final part showed a sensible increase on hardness (+ 40HB), that led to a revision of the heat treatment process. Heat treatment cycles have been evaluated by using FORGE NxT.

Benefits arised by the new approach

The implementation of the shaped ring rolling by using the usual trial & error approach should lead to 4 or 5 times longer development times for FELB, with considerable costs on real tests and equipment. The collaboration between FELB's know-how, Ing. Fioletti's experience and numerical design process experience provided by the Metal Forming Team of EnginSoft has led to a very fast and optimized development of product and related process, with the following advantages:

On FELB's side:

- radical innovation on the production processes, with implementation of shaped ring rolling and drastic reduction on development times (-80%) and costs (-50%) thanks to the numerical simulation;
- possibility to optimize the time plan, approving the equipment orders in less than one week and receiving them at the right time to start the production with an optimized preform (numerically developed during the waiting time for the delivery);
- validation about requirements of the rolling machine: forces and torques are below 70% of the nominal performance;
- optimization on material usage, with lower oversize material: weight reduction higher than 15% in comparison with the previous solution;

For the end customer:

- faster reply to quotation request, with evidences of the results obtained by the proposed process – possibility to develop co-design activities;
- higher performances on the final parts, due to the better compaction of the material and to the optimized fibres distribution provided by the shaped ring rolling process;
- relevant reduction on time and cost related to machining operations (up to -40%)

Conclusion and future developments

Thanks to the knowledge developed with FORGE NxT and the collaboration with EnginSoft, FELB was able to understand the advantages of the numerical simulation. Beside this case, FELB wants to use this approach in a very focused manner, referring to specific parts which will create new business, looking for for higher magnitude orders and increasing the overall efficiency.

Figure 5 – Real part during lamination (left) and at the end of the process (right)

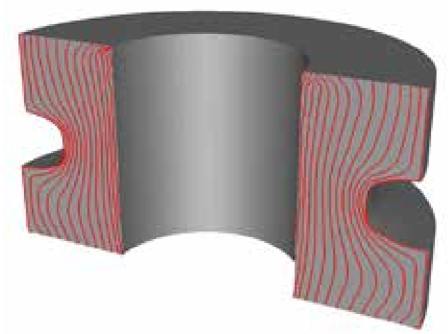


Figure 6 – Fibering on the final part

FELB is now developing similar offers to other customers and prospects, with high value proposal supported by the contribution of numerical simulation. This approach has already led to returns on business, as reported by the sales manager in FELB: "The obtained results confirm that the use of FORGE for the numerical simulation is definitely a good business card for our company, also talking on pure commercial terms. It allows us to differentiate in a very positive way, anticipating customers' objections and offering a sensible cost reduction and quality increase".
From the economic side, FELB and EnginSoft evaluated a Return Of Investment shorter than one year, by developing four or five parts similar to that one evaluated in the reported case, and taken into account the costs concerning hardware, software and employed people.
The support provided by the Metal Forming Team of EnginSoft was decisive to speed up the training phase, in particularly the shaped ring rolling process, which was developed and fine-tuned in less than one week.
Furthermore, FELB is spreading the use of FORGE also in heat treatment processes of forged and rolled parts, in order to perform a complete integration of numerical simulation in all the main company processes. The aim is to provide even higher quality to the end customers, which often operate in application fields, such as Oil&Gas and Energy, where metallurgical aspects (e.g. hardness, grain size, metallurgical phases) have a key role.

Fabio Fioletti, Felb
Federico Fracasso, Marcello Gabrielli - EnginSoft

I verified on the real case how the numerical simulation developed with FORGE was helpful to support the technical decisions, early in the proposal estimation. Thanks to FORGE, to mine and Ing. Fioletti's knowledge, and to the EnginSoft's support, we were able to offer a superior quality part, with a considerably increase on efficiency. I rediscovered in FORGE what I did in the past with the modelling clay, trying to understand the movement of the material. FORGE allows me to see in a very detailed way what happens to the material and the causes of specific defects, suggesting the best way to solve them. From a CEO point of view, I'm even now noticing interesting returns: we can now expand our business, in terms of batch size and/or complexity of the parts. Advantages on the efficiency on my production processes could be the key to establishing us on these new markets.
I would like to thank Ing. Fracasso, Ing. Capuzzo and Ing. Gabrielli of EnginSoft for their extensive knowledge and capability: they allowed us to transfer our real process on FORGE and to obtain crucial results in a very limited period, leading to important choices for the future development of my company.

Luigi Catalini, CEO FELB

EnginSoft "Metal Forming Team"

EnginSoft has been dealing with productive processes simulation for over 20 years with a group of engineers specifically specialized in metal deformation processes simulation. The activity carried out by this group ranges from the daily support to the over 70 Italian customers using Transvalor products Forge and ColdForm, to the training of those companies interested in undertaking simulation activities, to the performance of commissioned engineering works, up to the development of tailor-made models meeting customers' needs. The group competences extend from hot forging of steel and non-ferrous materials (brass, aluminum, titanium, …) for pieces of just few kilos, to open-die forging and circular lamination of steel for ingots of several tons, in addition to the simulation of cold rapid processes, such as bolts and screws and small parts forging.
The integrated approach provides the means to follow the material from the production phases of the billet/ingot (with the simulation of the continuous casting or of the ingot mould casting), to the heating, cutting, deformation, trimming and the heat treatment, carrying the stress accumulated in the part, as well as the porosity, the grain size and the related micro-structural phases. The objective of all these activities is to support the achievement of the best possible value of the simulation, measuring its benefits on the real improvement of the process/product quality: considerable results can be obtain in the first weeks of simulation activities, with a ROI within a year, thanks to the method setting up along the years and with continuous assistance.
For further information and to understand the benefits applied to your specific requirements:
ing. Marcello Gabrielli, EnginSoft - m.gabrielli@enginsoft.it

The Snamprogetti™ *SuperCups* Solution

The following article describes the results of the installation in two industrial facilities of the innovative Snamprogetti™ SuperCups technology. The design of the SuperCups was supported by a comprehensive CFD performance assessment carried out by Saipem and EnginSoft, that allowed to study the physical phenomena taking place into the reactor, in order to select the best SuperCups configuration and to confidently move to the test phase on field.
The improvements in mixing performances and residence time predicted by the CFD simulations were confirmed by the field data in terms of increased performance of the reactor.
The collaboration between Saipem and EnginSoft for the development of the SuperCups has been a continuous process of theoretical studies, CFD simulations and field testing that started in 2011 and culminated in 2014 with the first industrial application.
A more detailed description of the CFD study is available at:
http://www.enginsoft.it/newsletter/files/newsletter14-2.pdf

The performance of urea reactors can be improved by the application of the latest generation of internals: the Snamprogetti™ SuperCups. The proprietary design of such high efficiency trays is a further step ahead to approach the theoretical equilibrium conversion in the urea synthesis.
This paper presents the first world assessment of SuperCups performances achieved since 2014 with the installation and operation in the urea reactors of Borealis Agrolinz Melamine GmbH in Linz (Austria) and Fauji Fertilizer Company Ltd in Mirpur Mathelo (Pakistan).
The increase in the efficiency has permitted direct benefits to the overall day-by-day performances of the units, thus allowing a lower energy consumption and a reduced environmental impact.

1. THE SNAMPROGETTI™ SUPERCUPS

The fluid-dynamics of a urea reactor can be significantly improved by the introduction of the latest generation of internals recently invented and patented by Saipem.
The driving force for innovation has come from the continuous trend toward higher and higher plant efficiency with the aim to optimize the capital investment of the high pressure equipment, decrease the energy consumption and so reduce the environmental impact of plant operation.
The proprietary Snamprogetti™ SuperCups drastically increase the mixing of the reactants phases, respectively ammonia / ammonium carbamate and carbon dioxide, thus optimizing the product conversion rate in the reactor. The immediate benefit is the lower specific steam consumption requirement to decompose carbamate to CO_2 and NH_3 in downstream sections. Taking into consideration the necessity to minimize the pressure drop across the reactor, the improved mixing is obtained without any increase of compression energy for carbon dioxide.
This represents a further step ahead to get closer to the theoretical equilibrium conversion in the reactor.

2. DESIGN FEATURES AND WORKING PRINCIPLES

Urea reactors are non-ideal multi-phase plug-flow reactors ("PFR") type equipped with dedicated distributors for the reagents and a number of sieve type trays which consist of perforated plates that prevent back-flow of the heavier solution from the upper part downwards and favour the gas absorption in the liquid phase.

The innovative concept of the Snamprogetti™ SuperCups lies in the realization of a confined reaction space within the reactor tray geometry, namely the cups. They perform as a number of mixing units where ammonia is contacted with the gaseous CO_2 in small bubbles.

Once the reactants have swirled inside the cups, the mixed solution of product and non-reacted components is uniformly distributed on the upper part of the tray by means of the upper cup distributor. The outlet flow pattern ensures a further mixing of the solution coming from all the cups.

The peculiar behaviour of the SuperCups is characterized by a triple fluid-dynamic effect – Gas Equalizer, Mixer Reactor and Gas Distributor – which are described here below.

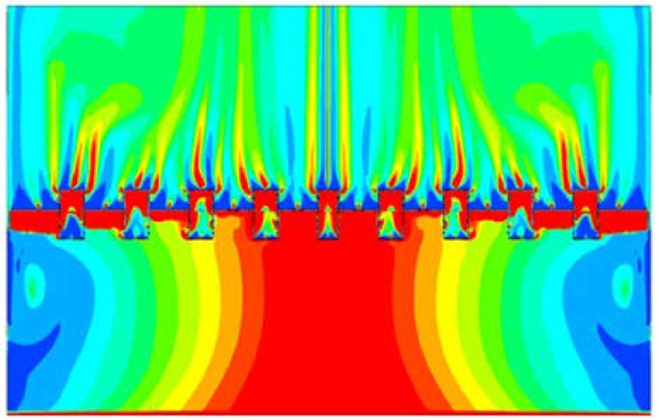


Fig. 1 - CFD of SuperCups tray showing CO_2 volume fraction

2.1 Gas Equalizer

The first effect of SuperCups is to uniformly distribute the concentration of the light phase reagent on the entire section of the tray. In this way, the gas-like bubbles moving upward "lose the memory" of the non-uniformity of the previous reaction stage and the non-reacted CO_2 can be evenly fed to each cup of the tray.

2.2 Mixer Reactor

The cups behave as a number of confined reaction volumes in which the multi-phase reagents – carbon dioxide and ammonia/carbamate – heavily swirl inside, thus reaching a high mixing degree. Each cup performs as a static mixer where the phases are strongly contacted (see Fig. 2).

Fig. 2 - Mixing reactor effect

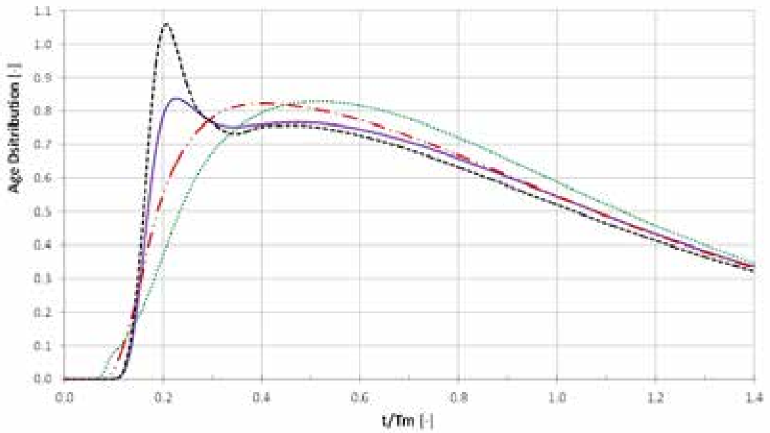


Fig. 3 - RTD curves for several types of tray design

In this way the SuperCups trays do not simply behave as gas distributors – like the existing commercial high efficiency trays – but perform as additional active reaction stages which can be modelled as a Continuous-Stirred-Tank Reactor ("CSTR") in series with the PFR of the stage.

2.3 Gas Distributor

The CO_2 phase forming the gas-cushion below the tray can be partially streamed inside the cups to create a mixer-reactor and partially distributed on the upper stage by means of dedicated holes. This split range is one of the most critical design parameters since it allows to customize the residence time distribution (RTD) curve of each reactor stage and to increase (or decrease) the CSTR (perfect mixing) or PFR (plug flow) behaviour according to the composition of each stage.

Fig. 3 plots, for several geometries, the RTD curves obtained by simulating a pulse injection of tracer at the inlet boundary and detecting the outlet concentration as function of time.

3. PERFORMANCE ASSESSMENT

After development of the innovative design, thanks also to the support of a comprehensive CFD study and of a mechanical assessment for constructability review, Saipem has decided to directly test on field the new technology before the commercial launch. Therefore, a joint collaboration agreement was set with two of its historical partners, Borealis Agrolinz Melamime International GmbH (Borealis) and Fauji Fertiliser Company Ltd (FFC).

The first performance assessments of the new SuperCups trays have been carried out in the year 2014 with a twin test in the urea plants, respectively, of Borealis located in Linz (Austria) and Fauji sited in Mirpur Mathelo (Pakistan).

This paper presents the results found along one year of operation after adoption of the new technology.

4. OPERATION AND RESULTS IN BOREALIS

Borealis is a Vienna-based leading provider of innovative solutions in the fields of polyolefin, base chemicals and fertilizers, operating production plants in 10 countries and employing 6,500 employees all over the world.

In Linz (Austria), Borealis operates production facilities for melamine with a capacity of 50,000 metric ton per year (MTPY), fertilizers like urea, complex fertilizers (NPK) and calcium ammonium nitrate (CAN) as well as technical nitrogen products like technical grade ammonium nitrate and guanidine carbonate (1,500 MTPY in total), making Borealis the second-largest producer of melamine and market leader in fertilizers in the Danube region.

The urea plant in Linz is based on Snamprogetti™ process and currently produces approximately a total of 400,000 MTPY of urea.

The urea unit is supplied with ammonia and carbon dioxide produced on site and feeds two melamine plants (one high pressure and one low pressure process), thus forming a highly integrated plant network.

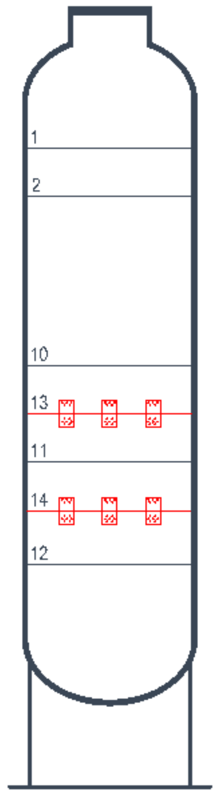


Fig. 4 - Sketch of modifications implemented during SD-2014

4.1 Installation of SuperCups during 2014

During plant shut-down in April 2014 (SD-2014), two SuperCups trays have been installed in the urea reactor as additional items.

The new trays have been designed keeping the same sector assembly of the original ones with a central man hole cover and have been positioned between the existing trays 10 – 11 and 11 – 12 where spacing was higher than the upper trays, resulting in an evenly spaced tray alignment as shown in Fig. 4.

No other modification relevant to the synthesis section of the urea plant was carried out during the plant shut-down.

4.2 Plant Performance after Installation of SuperCups

The performance assessment has been carried out by monitoring the high pressure loop and medium pressure steam network of the urea plant in a wide time frame before and after the installation of the trays.

The first remarkable point in the evaluation of the plant performance after the installation of the new trays comes from the evidence of increased maximum achieved plant capacity.

Table 1 shows the capacity increase after SD-2014 calculated from maximum plant load for an increasing number of subsequent days.

Table 1
Plant Capacity Increase after installation of two SuperCups during SD-2014

	5 day average	21 day average
Capacity Increase [%]	**2.2**	**1.9**

4.2.1 Urea Reactor Performance

Another part of the performance assessment consisted in the analysis of the urea reactor outlet stream composition before and after the trays installation.

Analytical data of the reactor outlet composition shows that the installation of two SuperCups has moved the average operating area of the urea reactor toward higher conversion rates at increased plant loads.

In addition, in order to provide a quantitative indication of the reactor performance at different plant loads, the analytical data relevant to operation before and after the trays installation has been rearranged to obtain the mean residence time (MRT) of the reactor.

The MRT in the reactor has been calculated as ratio between the overall reactor outlet flow rate and the reactor volume; the reactor outlet flow rate has been calculated from plant load and urea mass fraction resulting from reactor outlet analysis. In this way, two sets of data of mean residence time were obtained, describing the reactor performance as a function of plant load before and after the trays installation.

Fig. 5 plots, for each plant load, the difference between the mean residence time before and after the installation, defined as follows:

$$MRT\ Increase\ [\%] = \frac{MRT_{After} - MRT_{Before}}{MRT_{Before}} \times 100$$

It is remarkable that the MRT Increase is higher for higher plant loads, thus showing that, when the reactor is operated in the limiting conditions of very high capacity, the improvement of fluid-mechanics permits to effectively enhance the performance of the reactor.

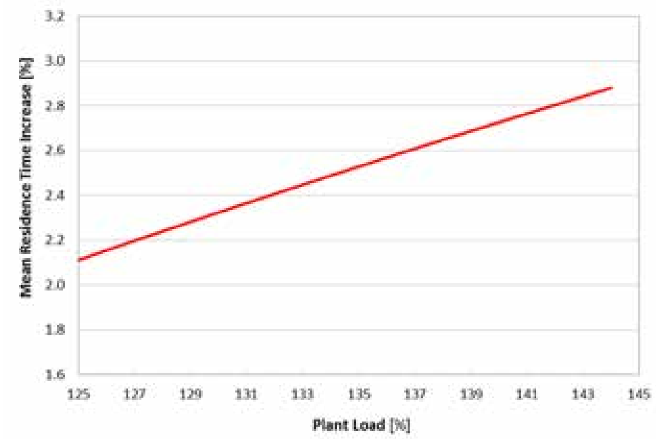


Fig. 5 - Increase of MRT after SuperCups installation vs. plant load

4.2.2 Steam Balance

The evaluation of the overall steam duties and consumption figures is based on plant data collected in several series before and after the plant shut-down.

Noticeably, the plant load was increasing after the installation of the SuperCups, while the overall medium pressure steam consumption stayed the same, resulting in a decrease of the specific heat duty by 2.3 % after SD-2014 at the plant capacity of 142 %.

Compared to the energy consumption prior to installation, this

lower specific energy consumption results in a net saving of 1.3 t/h of medium pressure steam at maximum plant load.
Based on the specific steam saving found in the assessment, it was also calculated that the theoretical CO_2 conversion increase at reactor outlet is + 0.74 % after the installation of two SuperCups.

5. OPERATION AND RESULTS IN FFC

Fauji Fertilizer Company Limited (FFC) is one of the largest urea manufacturers of Pakistan. FFC operates three large scale ammonia-urea complexes with aggregate annual urea production capacity of over 2.4 million metric tons.
FFC has vast experience of Snamprogetti urea synthesis technology operation. FFC has been part of the technology evolution process through collaboration in testing and development of Omegabond Stripper technology and testing of SuperCups urea reactor trays. FFC Plant-III has been one of the earliest testing grounds of both these technologies.
The plant of Mirpur Mathelo was commissioned in 1981 with design urea production capacity of 574,000 MTPY. FFC acquired this plant from Government of Pakistan through privatization program in 2002, and carried out a major capacity revamp in 2008 to increase plant capacity up to 718,000 MTPY (125% of the original nameplate capacity).
The urea reactor originally contained 10 sieve trays of old design. During the capacity revamp, 05 more sieve trays of the latest design were added below the original trays.

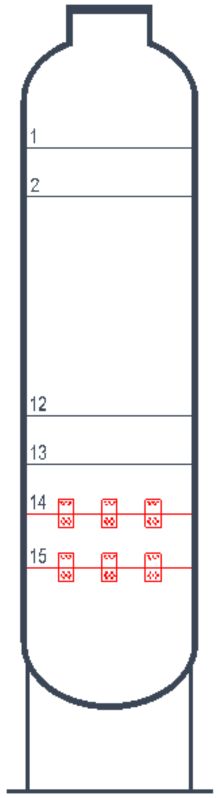


Fig. 6 - Sketch of modifications implemented during TA-2014

5.1 Installation of SuperCups during 2014

During last plant turnaround in September 2014 (TA-2014), two of the lowest bottom sieve trays were dismantled and replaced with SuperCups trays (see Fig. 6). In order to install these trays on the existing support system, a modified design consisting of a C-shaped ring (installed at site) was employed to position, adjust and fix the new trays. Installation time of the new trays was somewhat higher than the sieve trays primarily due to relatively higher weight of the system (ring and tray) and for constraint in cradle movement inside the reactor. The activities were anyhow performed on time.
No other modification was implemented during the plant turnaround – that was mainly dedicated to ordinary maintenance.

5.2 Plant Performance after Installation of SuperCups

Detailed evaluations have indicated improvements in the reactor performance and HP stripper steam consumption after the installation of SuperCups trays. Urea reactor conversion has in fact increased with consequent decrease in the HP stripper steam consumption. It is expected that installation of a complete set of SuperCups in place of the sieve trays shall yield significant benefit in stripper steam consumption.
The first important remark is given by the fact that, after the installation of two new trays, the urea plant has been operated at higher loads. Fig. 7 shows the plant capacity along the whole 2014 up to August 2015 as percentage of original design capacity. The average load after SuperCups installation has increased by about 4 % with respect to the period before TA-2014, reaching peaks higher than 140%, thanks to the availability of additional natural gas.

5.2.1 Urea Reactor Performance

The starting point of the performance assessment consisted in the evaluation of urea reactor outlet stream composition throughout the entire 2014 and up to August 2015, before and after the trays installation.
The map of urea reactor analyses show that the replacement of two sieve trays with two SuperCups has mostly shifted the operating condition of the reactor toward higher conversion rates at increased plant loads.
In addition, based on the reactor analyses collected at different plant capacities before and after TA-2014, the mean residence time has been calculated as the ratio between the reactor outlet flow rate (depending on analyses) and the reactor volume (as per equation 1). The MRT Increase after the installation of SuperCups trays has been plotted versus the plant load in Fig. 8.

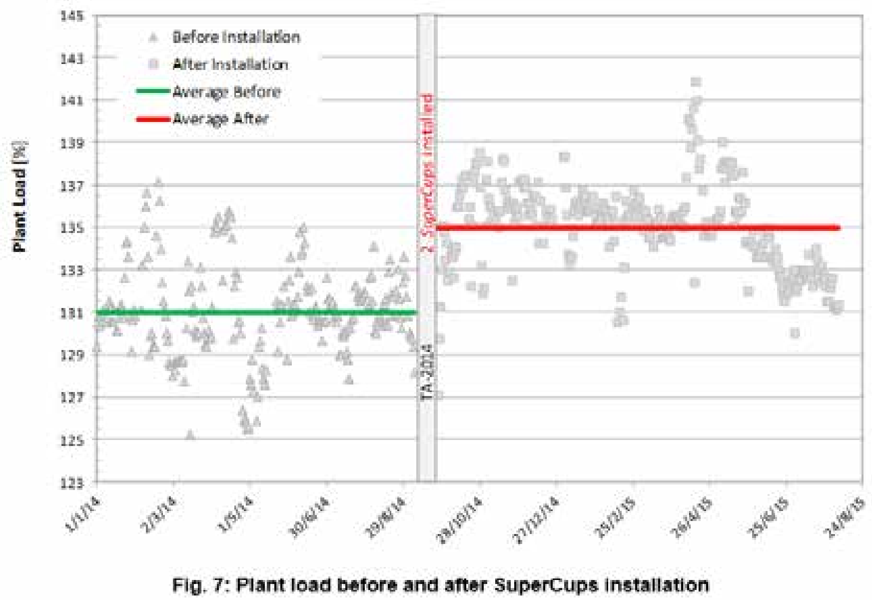


Fig. 7: Plant load before and after SuperCups installation

Fig. 7 - Plant load before and after SuperCups installation

The MRT Increase grows steadily with the plant capacity and becomes more significant at high loads. In other words, for high capacities, the reactor volume tends to be limiting for the conversion and the improvement of fluid-dynamic patterns through the SuperCups allows to overcome this limitation and to operate the reactor at increased capacity with enhanced conversion.

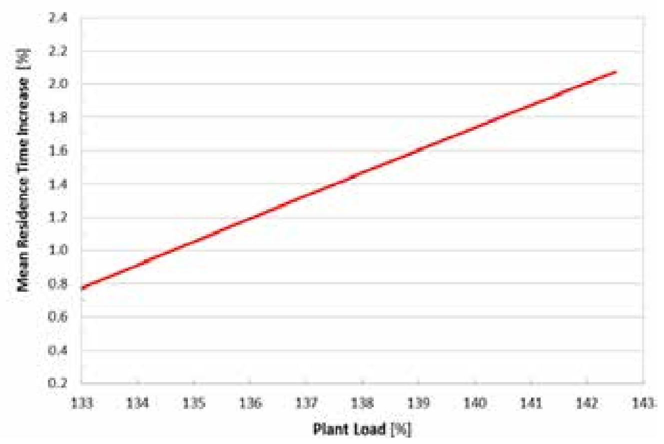


Fig. 8 - Increase of MRT after SuperCups installation vs. plant load

5.2.2 Steam Balance

The evaluation of the overall steam duties and consumption figures is based on the plant data collected on a number of dates before and after the plant turnaround of 2014.
Results show that the specific heat duty of each steam user is lower after the installation of the SuperCups trays and, in particular, the reduction of overall specific duty of medium pressure steam (MS) users is ranging between 1.1 ÷ 1.9 %, corresponding to a direct saving of MS steam from 0.9 t/h to 1.6 t/h.
Obviously the greatest contribution is to be attributed to the urea stripper, however, the reduction of MS consumption in the M.P. Decomposer (via booster ejector) is anyway significant.
Based on the specific steam saving mentioned above, it has been calculated that the theoretical CO_2 conversion increase in the urea reactor ranges between 0.30 % and 0.54 %.

6. CONCLUSIONS

As of today, the SuperCups have been comprehensively tested in two industrial facilities before the commercial launch.
The performance of these innovative trays has been assessed by monitoring the plant operation over a wide time frame. The laboratory analyses, the steam balance and the process evaluation of High and Medium Pressure sections all lead to the evidence of an enhancement in the performance of the units that, after the installation of SuperCups, have been operated at increased plant loads with higher reactor yield and lower specific energy consumption.
In addition, at maximum plant capacities, the reactor volume starts to be limiting and the improvement of fluid-dynamic patterns given by SuperCups allows to push the reactor conversion upwards thanks to the higher mixing efficiency of reactants and to the increased mean residence time. These operational finding results are even more significant considering that the extent of modifications was limited to two trays only for each unit.
In conclusion, based on its proven features, the SuperCups represent a completely innovative reaction device for urea synthesis that can be advantageously applied to design a new generation of urea reactors as well as to improve the performance of existing equipment in retrofit design.

This article was presented at: Nitrogen + Syngas 2016 International Conference (Berlin, 29 February – 3 March 2016)

ELMAR LADSTÄTTER, ALEXANDER GUSENBAUER
Borealis Agrolinz Melamine Int. GmbH
Linz, Austria
MUHAMMAD ASLAM, ASHRAF KHAN
Fauji Fertilizer Company Ltd.
Mirpur Mathelo, Pakistan
UGO AVAGLIANO, LINO CARLESSI - Saipem S.p.A.
Milan, Italy

Avagliano Ugo - Ugo.Avagliano@saipem.com
Serrafero Alberto - Alberto.Serrafero@saipem.com

Multiphase flow simulation applied to predict coke capture in a pipeline for Petrochemical industry

The petrochemical industry represents millions of tons of products per year and it is heavily present in modern life, with wide applications in diverse sectors such as clothing, automobile, health, and more. In the olefins plants, equipment with great importance are the pyrolysis furnaces, which are responsible to thermally crack naphtha, LPG, ethane and recycle streams to produce the main products of the petrochemical industry like ethylene, propylene, hydrogen, gasoline and pyrolysis residue.

One of the main problems of the pyrolysis furnaces is the gradual formation of a complex created from carbon and hydrogen, called coke, in the coil walls of the furnaces. The equipment runlength is defined by the maximum temperature that the walls of the radiation coils reach during operation, caused by the coke deposition onto the inner walls. Normally, the operation with high severity increases the production of the main product ethylene, but also increases the formation of coke and reduces the furnace runlength due to thermal and flow area restrictions. Therefore, the performance of the pyrolysis furnaces is critical for the economic performance of an olefins plant. Hence, in order to eliminate the coke from the furnaces, it is necessary to interrupt the operation and submit the radiant coils to a process of decoking which is the controlled burning of coke, using a mixture of steam and air. Part of this coke does not burn and is mechanically dragged from the radiant coils accumulating upstream of the main Transferline block valve. When the furnace is put back into operation and aligned to the process (main Transferline), it flows as a plug of coke particles to the Primary Fractionator and Quench Oil Loop, resulting in filter obstruction (sometimes damage), erosion problems, obstruction of Quench Oil Heat Exchangers, and even reduction in the plant capacity. Thus, the most efficient way to solve this problem is avoiding the deposition of coke, transporting it directly to the decoking pot during the decoke operation.

This article explains how BRASKEM used ANSYS computational fluid dynamics (CFD) software to numerically replicate the geometry of the transfer line of the pyrolysis furnaces, evaluating the impact of the operational conditions and geometry on the velocity and pressure profiles and mainly on the prediction of the retained coke along the transfer line. The project aimed to evaluate the multiphase flow (gas and coke) in the transfer line of the pyrolysis furnaces during the decoking process to improve the efficiency process with minimal change in existing geometry. The mesh was generated in ANSYS Meshing with 650 thousand tetrahedral and prismatic elements. Braskem ran ANSYS Fluent simulation assuming steady state and isothermal flow. The flow was treated as multiphase with Euler-Lagrange one-way coupling approach and SST turbulence model. Each simulation took 4 hours to run on a high performance computing (HPC) platform with 12 cores.

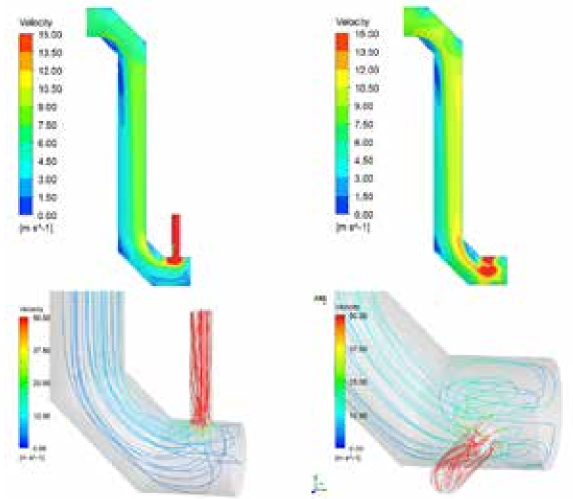


Figure 1 – Velocity profiles and streamlines for two different geometries in the transfer line: (a) vertical outlet pipe and (b) horizontal outlet pipe

The CFD analysis allowed BRASKEM engineers to identify coke deposition behavior under different operational and geometric conditions. As expected, higher gas flow rates resulted in lower retained particles percentage in the transfer line. However, for the particle size distribution considered, a small standard deviation was observed in the retained percentage for the larger particle diameters (3250 and 4000 μm), indicating that, even with higher flow rate, the larger particles tend to remain in the domain. This results suggest that the system requires geometric modifications for efficient particle removal from the transfer line, and that the operational conditions have a limited impact on the retained coke percentage. Figure 1 illustrates the velocity profiles for two different geometries.

The simulation revealed significant areas for improvement of the original design. The Figure 2 shows the retained coke percentage for two different geometries by changing the outlet pipe lay out. Due to some geometry restrictions, the changes in transfer lines were very subtle. The proposed modification consists of the rotation of the outlet pipe in 90° and it was observed that this simple modification reduces the coke mass deposition in 90% in comparison with the vertical pipe case. The suggested geometry promotes greater velocity in the bottom of the pipe and consequently removes larger amount of coke due to the reduction of recirculation and low velocity zones.

In order to better predict particle behavior inside the line, a Fluent-Rocky one-way coupling simulation was performed, as it considers particle-particle and particle-walls interaction in a deterministic mode, in spite of the larger computational cost of this technology due to the greater number of particles (over a day depending on the number of particles tracked). This simulation showed that the coke accumulates at the bottom of the transfer line, and at a certain position upstream of the main Transferline block valve, confirming the phenomenon observed in the actual industrial process. Figure 3 shows the Fluent-Rocky coupling simulation result, showing the particle deposition locations.

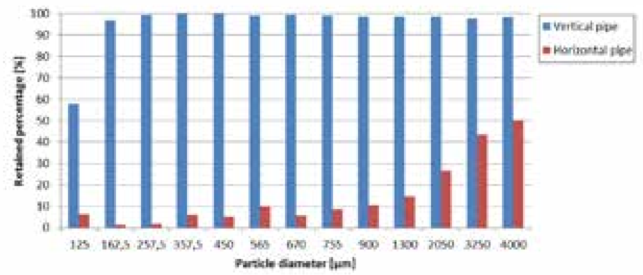


Figure 2 – Particle retained percentage for two different transfer line geometries

The CFD or CFD-DEM models explained fairly well the impact of operating conditions and geometry changes on coke accumulation inside the transfer line. Furthermore, the use CFD-DEM coupling should be done for final quantitative verification, given a particle size distribution to better represent the physics involved.

This application demonstrates that fluid dynamic simulation can be used to predict multiphase flow behavior in order to avoid undesirable operational problems generated due to coke accumulation in the transfer line, saving maintenance costs and ensuring safer operation of the equipment with greater efficiency.

Marcus Reis, ESSS
www.esss.com.br

BRASKEM is supported by ESSS, ANSYS elite channel partner in South America.

For more information:
Luca Marcadent, EnginSoft - l.marcadent@enginsoft.it

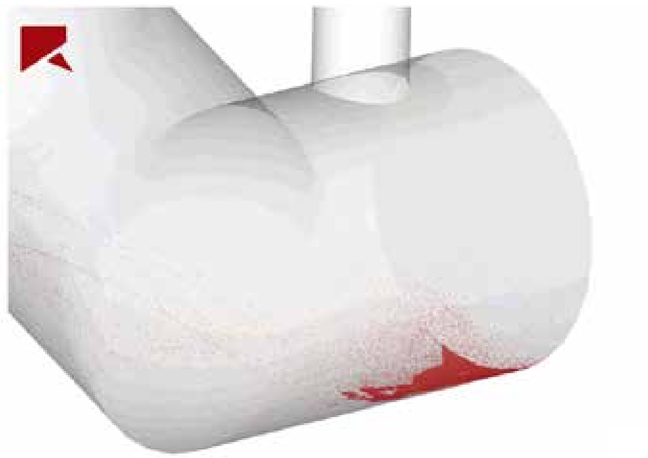


Figure 3 – Fluent-Rocky coupling result for the vertical outlet pipe geometry

The SACMI Group, cutting-edge Technology for Industry

The SACMI Group have been looking for a new software that is created around designers, production designers and quality managers for quite a while and were specifically interested in a tool to identify and eliminate the problems caused by error propagation in the production process.
Therefore, when EnginSoft introduced CETOL 6σ, we immediately started a pilot project in collaboration with EnginSoft to highlight all the benefits. The project lasted for one month and during this time we have been positively supported by EnginSoft technicians to achieve an extremely satisfying result. We can state that the Derivative Based Approach used by CETOL 6σ to solve the static combination of tolerances, has allowed us to achieve results in just a few seconds, in comparison with other more expensive computational methods (as the Monte Carlo one). Furthermore, we have noticed how CETOL 6σ's post-processor has been designed to highlight thecritical factors and see how a change to them can effectscrap rates in just a few steps. This allows us to intervene and reduce production scrappage.The optimum corrective measures (both in terms of quality and cost) were identified by using the the information produced by CETOL 6σ's advanced graphical output. For example, the statistical distribution of any critical functional requirement can be visualized alongside the induvial contribution from each component effecting any non-conforming requirement. The project ends with the automatic generation of a detailed report, useful for the production designer and the quality manager. Practically speaking, CETOL 6σ has proved to be a very powerful technology for parallel engineering; it has allowed the SACMI Group to save time in the design and prototyping phases while minimizing scrap in accordance to the ever more widely diffused lean manufacturing.

Eng. Stefano Bergami
Technical Manager, SACMI

SACMI is a multinational group that manufactures machines and complete plants for the Ceramics, Packaging (Beverage and Closures&Containers), Food and Automation industries. They are a recognized worldwide leader due to their application of innovative technology, their well-established position in the international market and their commitment to research that ensures consistent top quality and service to the client. The SACMI Group has over 80 companies in 28 countries, production plants, distribution firms and service companies controlled by the parent company in Imola. A far-reaching presence that explains why over 89% of Group sales come from exports. The SACMI Group provides efficient assistance and services all over the world thanks to the company's global network which offers fast, efficient, direct service to every continent. The SACMI Group have become a major international plant engineering firm in key sectors of the economy by: maximising investment in research, unwavering promotion of technological innovation, conscientious attention to product and service quality, effective responses to real world market needs, extreme flexibility in taking advantage of technological integration and carrying out the integration of operations for widely different industries.

Cetol 6σ in design
CETOL 6σ is a software technology able to evaluate the effects dimensional tolerances have on the various functional requirements that a complex product must satisfy. It is important to ensure that manufacturing variation doesn't affect a product's assembling and/or operation. Thus effecting profits and market confidence in the brand.

In the project undertaken, an analysis was conducted on a rotary machine for cap moldings. This was done to avoid 'sticking events' during the molding process (a performance requirement to avoid machine stoppage) and to strictly control the alignment among the molding parts (ensuring the high quality of the final product).
CETOL 6σ allowed the easy undertaking of a 3D tolerance analysis, in a quick and intuitive manner. In comparison, a 3D tolerance analysis made by hand calculation or through excel sheets would be complex, time consuming and prone to mistakes. Moreover this activity has a positive impact on many company's functions, going well beyond the results of a single analysis, bringing significant competitive advantages in the present market.

Thanks to the analysis, the technical office was able to generate 2D drawings for the production department, with optimized tolerances that ensured the product requirements are always satisfied (assuming that parts respect their assigned tolerances), and the costs are minimized. CETOL 6σ has applied additional accuracy where strictly necessary while applying cheaper, more general tolerances where unrequired. CETOL 6σ also made the application of GD&T (Geometric Dimensioning & Tolerancing) easy for the production of 2D drawings by applying a rigid syntax that is non-ambiguous and unopened to misinterpretation. GD&T therefore gives the advantage of using a clear, shared language among design, production and quality departments (also strictly outlining the correct dimensional tests), which helps avoid costly misunderstandings between internal departments as well as external companies.

All dimensional controls are decided based on the results and limited to the verification of few critical measures and their tolerances with the greatest impact on product requirements. This leads to the drafting of a dimensional inspection plan that collects

only the most important information on the product's quality from a statistical point of view. This omits unnecessary geometric complexity and thus streamlines the number of dimensional controls. In SACMI the quality surveys are targeted to extract data whose added value is to be used for the verification of virtual prototypes. In fact, the model created for the calculation is reused to evaluate the impact of supplier delivering "out of tolerance" parts. The "acceptability of a derogation" is not evaluated by a long and expensive prototypal campaign or further mathematical calculation, but is virtually assessed by entering the recorded measurements into the CETOL 6σ model to view the updates instantly.
In SACMI the application of the propagation management method of dimensional and geometric tolerances, supported by the use of such tools as GD&T and CETOL 6σ, fully integrated into their CAD software, has allowed the identification and fixing of potential sources of failure from a virtual prototype, thus avoiding later costly manufacturing failures in the product development cycle.

Sigmetrix' Impressive New Version of CETOL 6σ Tolerance Analysis Software Has Arrived!

CETOL 6σ Tolerance Analysis Software is More Intuitive and Easier than ever! New version of CETOL 6σ Tolerance Analysis Software (v9.0) Emerges in Style with Intuitive Workflow and Enhanced Ease of Use for Engineers of all Expertise Levels

CETOL 6σ tolerance analysis software is tightly integrated within CAD systems to provide a seamless environment. Users of all expertise levels will appreciate the multiple levels of assistance now available, ranging from enhanced tool tips to the intuitive Advisor messages based upon what step of the process the user is in. These Advisor messages show the user what they need to do when they need to do it. The context sensitive, state-of-the-art help system has been completely revamped and now includes practical tutorial videos to refresh your understanding of certain steps and techniques.

The entire UI was evaluated end-to-end so that the resulting interface was optimally aligned with the most common user workflows. Specific emphasis placed on ease-of-use for model creation, obtaining answers, validating results, and updating design requirements in the CAD models and drawings. Not only does it look updated and modern, but the layout is now optimized to make tolerance analysis definition easier.

Continuing with the focus on usability, Assembly States have been added to allow for greater flexibility in defining which components are to be included in each analysis. Engineers don't have to redo previous analyses from scratch when updating their design, and changes made to tolerances within CETOL are automatically updated in the CAD models and drawings.
CETOL 6σ v9.0 provides expert insight through unique visualization capabilities. The results and reports provide the entire development team with a richer way to communicate and help bridge the gap between design, manufacturing and assembly departments.
For more information:
Enrico Boesso, EnginSoft - e.boesso@enginsoft.it

Optimization of the production process of an automotive die-cast component with MAGMA5

Cold-chamber die-casting is a particularly advantageous process for high volume production, due to its short production cycle and low cost. Currently, half of the world's production of castings in light alloy are manufactured with this technology while continuous improvement in quality is broadening its application to automotive components.
This trend is clearly shown in this article, which describes the development phases in the production of a Magna component with high functional performance demands.
In particular, the benefits of using the MAGMA5 process simulation in the design flow are presented. The process allows designers of Studio DSM to validate complex solutions in advance and reach the expected quality targets, while conforming to short development times and limited cost. This is done for both the industrialization and the operation of the new equipment.

Introduction

A truly optimized production process consists in maximizing the product quality but also in minimizing the related production cost. Invariably, these objectives are in conflict, meaning considerable design effort that could become a real technological challenge.
The case study described refers to an automotive aluminum die-cast component, produced in a cold chamber (Figure at the top of the article) and reflects the latest automotive trends: weight reduction, cost reduction and increased reliability.
In fact, the greatest economic advantage for high pressure die casting are the high production volumes while the reached quality of the product is often lower in comparison to that one obtained through alternative production methods. With this in mind, the design development process that has allowed Studio DSM designers to improve the original equipment is presented in terms of both increased product quality and profitability.

STUDIO DSM
DIECASTING DESIGN

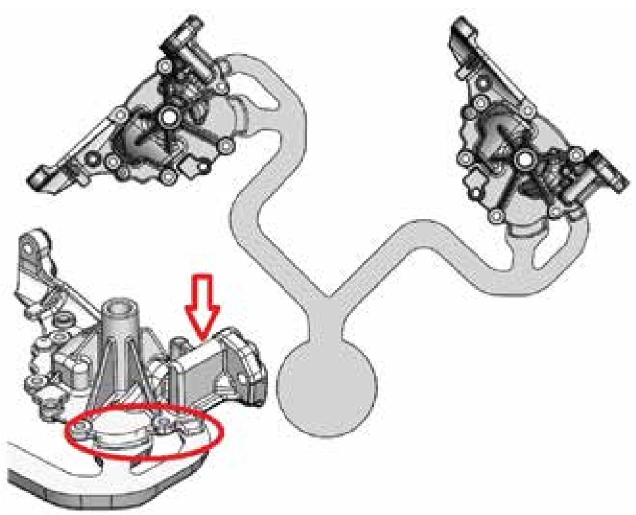


Figure 1 - initial hypothesis (V01)

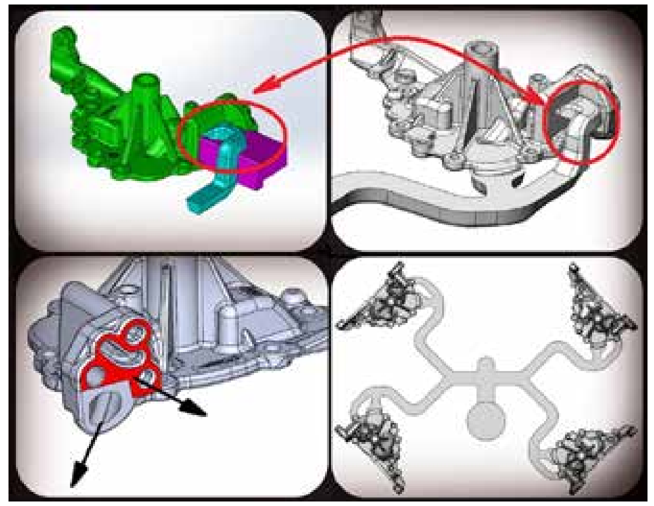


Figure 2 - Final hypothesis (V02)

Design Guidelines

Magna, who commissioned the project, are a global leader in the supply of automotive components who have made quality and customer satisfaction the core of its business.
The component in question is a cover that ensures maximum operational reliability in both structural strength and the hydraulic seal (preventing oil leakage).
The component consists of an oil-pipe, which is obtained by machining a thick folder that provides a perfect hydraulic seal, i.e. it must not have any defects when withdrawing to raw state.
The high volume of production expected for this part, together with possible cost savings, makes the high pressure cold-chamber die casting process particularly suitable for the production of this part.
Galba S.r.l the primary contractor for the supply of this machined component has identified, in accordance with Alpress S.r.l foundry, a reference mold with two cavities.
The high quality features required have persuaded engineers to conduct a virtual preliminary study of the mold's production process, in order to identify in advance any critical problems for the part's internal integrity. Studio DSM has been involved in the realization of the mold, taking care of any single detail, verifying its performance using MAGMA5 for the process simulation.

Design

The mold initially considered was one with the minimum number of production cavities and the reference comes from the shape of the reference product's CAD geometry. The defined mold division line lies on a plane and allows the simplification in the construction of the casting system which can be entirely defined on a single division surface, directly adhering on the replica cavity on the wider perimeter free from obstacles and on the closest one to the thick folder, with a total of two gating for each cavity (Figure 1).
Once the casting system was defined, it was possible to proceed with verifying the expected performance using MAGMA5 to evaluate the mold robustness while varying the process' parameters.
The virtual analysis of the initial configuration highlights some considerable critical points, concerning both the filling and the solidification phase: on the one hand the filling dynamics showed a marked imbalance between the two gating that serve each cavities. In particular, the preliminary analysis of the filling of the cluster deliberately freed from overflows, revealed no preferred areas of gas accumulation for venting which have, on the contrary, a uniform distribution with critical concentrations for gas entrapment. Additionally the solidification dynamics have not provided satisfactory results due to a considerable risk of isolated shrinkage areas detected in the manufactured thick folder which must ensure hydraulic seal.
One solution to the porosity problem could be to place a squeeze-pin close to the massive area, but the high concentration of gas present there could not guarantee a good effectiveness of this solution.

It was therefore decided to study a targeted casting system on the component's massive area, to reduce the shrinkage porosity to a minimum while limiting the residual air. This action implied an additional mechanical movement for each figure, determining a higher production cost and influencing the mold maintenance. In order to reduce the production operating cost, it was then decided to add two further cavities, leading toward a four-cavity mold (Figure 2).
The new casting has not only reduced the critical aspects for internal integrity on the massive area, as visible by comparing the "porosity" results (Figure 3), but also concentrate the air in a peripheral area of the component allowing for easy evacuation.
Once a satisfactory casting system was identified, the design is completed by defining an effective system to exhaust the residual gases and an appropriate thermoregulation system (Figure 4).
The related simulations are then used to virtually verify dimensioning of the entire equipment and the process parameters. The achieved results confirm the initial design expectations (Figure 5) and have permitted the direct move from the design to the manufacturing phase.

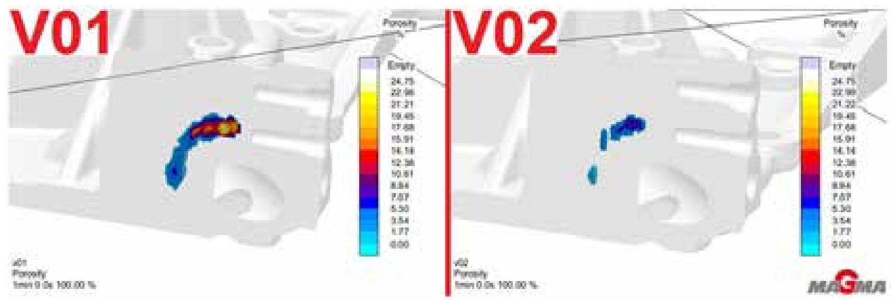


Figure 3 - Porosity v01 VS v02

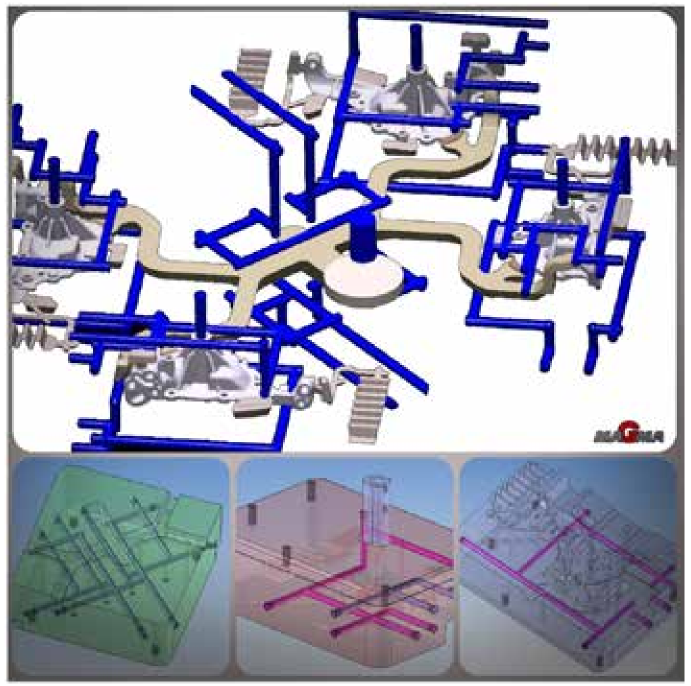


Figure 4 - Thermoregulation and venting system

Sampling

Before starting equipment production, the possible design choices are presented to the project partners involved. The strong impact provided by the post processing tools integrated within MAGMA5 made it possible to adopt the new mold construction in full agreement from all the parties.

With great satisfaction, the samples proved to be problem and compliant free from the very first batch.

Finally, the customer required an in-depth test on some sample castings using 3D computed tomography; this technique of non-destructive testing has the advantage of allowing the detection and measurement of defects in internal integrity (shrinkage porosity and cavity by air entrapment); although repeated sample checks have revealed some small non-systemic defects close to the thick folder (with positions and inconstant extensions probably due to air entrapment), they have also confirmed the health of the thick folder and the consequent possibility of mechanical machining, minimizing the risk of an imperfect hydraulic seal of the operating component.

The sampling phase of the new equipment was satisfactorily concluded with the setting up of the process parameters, practically corresponding to those defined for the final simulation, and with the conformity verification of the produced samples, therefore ready for the market.

Conclusions

This article has described the design flow that has led to the definition of the process for the production of a cold-chamber die-cast in, intended for the production of a Magna automotive component, characterized by high technological performance and high volume production.

The various phases of industrialization were presented, starting from the initial design guidelines until the sampling, giving particular emphasis to the benefits obtained from the use of the process simulation with MAGMA5 code.

An in-depth study of each production phase with the aid of virtual techniques has allowed Studio DSM engineers to create a fruitful synergy combining of their experience with the advance testing of their design concept.

In particular, the most interesting result has consisted in obtaining the demanded quality at a lower cost than the foreseen one, in a very limited development time, fully satisfying the customer.

In conclusion, it is important to consider the statement by Stefano Chiarello, the designer of Studio DSM who mainly worked on the virtual analysis of the process: "I have realized the usefulness of the software for decision-making support, especially when experience was suggesting me that the initial assumptions taken into account could be risky'.

The Company: Studio DSM

Studio DSM proposes itself to high pressure die-casting foundries of light alloys and zama as ideal specialized partner in mold design, able to manage all phases of equipment implementation, including the construction and the setting up of the productive process.

Studio DSM team of technicians takes advantage of forefront technologies, guaranteeing its customers the necessary technological innovation to compete at a global level.

Mauro De Santi, Stefano Chiarello, Studio DSM
Giampietro Scarpa, Lorenzo Trevisan, EnginSoft

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

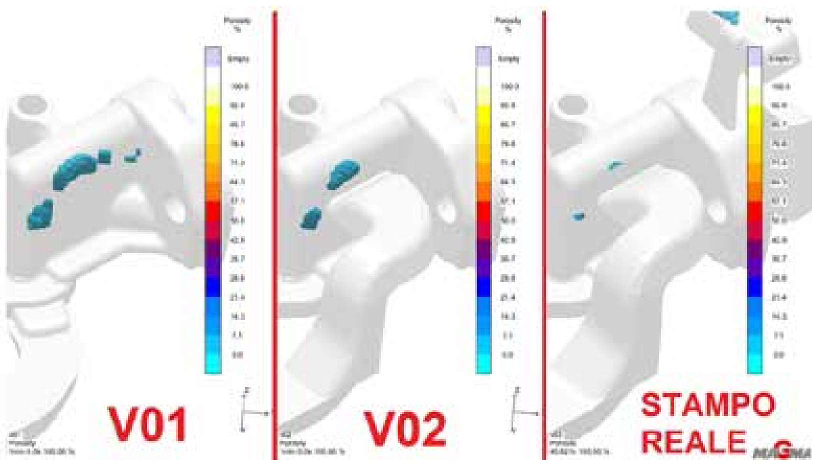


Figure 5 - Porosity

Multi-Disciplinary and Multi-Objective Optimization of a NASA Unconventional Aircraft Concept

The Over-the-Wing-Nacelle is an unconventional aircraft concept, which design cannot rely on accumulated experience and databases. An explicit multi-objective approach has been used to solve such multidisciplinary problem, aiming to optimize the structural and aerodynamic performance

Langley Research Center (LaRC) is the oldest of NASA's field centers, located in Hampton, Virginia, United States. The Center currently devotes two-thirds of its programs to aeronautics, and the rest to space. This research activity was carried out at the Systems Analysis and Concepts Directorate (SACD).

1. Background and application overview

The Over the Wing Nacelle (OWN) is a concept by Andrew Hahn from NASA Langley Research Center, which is promising to be a new solution for improving the aerodynamics in the transonic flight condition. The main differences between the OWN and a conventional aircraft are the location of the engine over the wing and the extension of the leading edge of the inboard section of the wing. Although this concept has been already optimized from the aerodynamic and performance points of view, its preliminary design requires further investigation by taking into account a more complete scenario. The structural assessment was not performed and therefore, a multidisciplinary design analysis is necessary in order to evaluate and propose a possible airframe baseline.

The wing box structure is the component that needs a more detailed investigation of its structural dynamics, because of the unusual engine location, as well as the extended wing leading edge, which may affect the dynamic and aeroelastic behavior. Specifically, the aeroelastic response was considered as a critical point for the flutter insurgence. Therefore, a multidisciplinary environment has been proposed, comprised of the mission, aerodynamic, structural and aerostructural assessment carried out by specific tools. The purpose of this optimization analysis is to drive the preliminary design towards a set of better designs from a global point of view. Indeed, in this design phase, designers cannot expect to obtain the exact answers about design variables and response variables, but they can look for the best direction to follow. Therefore, the analysis fidelity should reflect this reality and should include all of the main analyses, maybe making approximated models preferred over performing very detailed simulations. In this study case, the main design requirements according to the Federal Aviation Regulation for transport aircraft are taken into account early in the design process, and thus the necessary analyses for evaluating such response parameters need to be performed. In order to include all these analyses in the iterative process for the MDO of the OWN while keeping the total effort under control, both high-fidelity and approximated models are used. The level of fidelity in this application aims to correctly evaluate the overall physical behavior of the aircraft and the response trend when design variables change, but at the same time keeping low the computational effort.

2. The Multidisciplinary Optimization problem for the OWN

The MDO problem for the OWN accounts for the disciplinary behaviors which mostly impact the overall design, such as structures, aerodynamics, aeroelasticity, mission performance, stability, and control.

Objectives to be optimized

Traditionally, the Take-Off Gross Weight (TOGW) is used as optimization objective, because it correlates highly to the overall Lifecycle cost of the airplane.

The TOGW results from the sum of the three main weight factors, which are the Empty Weight (EW), the Operating Weight (OW) and the Fuel Weight (FW).

Since the purpose of this work is to explore all possible design solutions and then select a few ones for the next design phase, the idea is to push the design and optimization process to do it by selecting multiple objectives representing contrasting discipline requirements.

For this reason, the two components EW and FW are considered as objectives of the optimization, because they represent two opposing behaviors, respectively, the structural lightness aiming for a low-aspect-ratio design, and the fuel economy aiming for an high L/D design (i.e., high aspect ratio).

Design variables

In this application both structural and shape variables are used in order to properly affect all of the involved disciplines. The former are necessary for the airframe optimization and the latter for the external geometry optimization.

The list of the design variables is reported in Table 1.

	WING AND TAIL	FUSELAGE
Structural Variables	Skin Thicknesses	Skin Thicknesses
	Spar Web Thicknesses	Stringers Areas
	Spar Cap Areas	Floor Thickness
	Stringers Areas	Floor Stringer Areas
	Stringer Number	Stringer Number
	Rib Number	Frames Number
	Materials	Materials
	LE and TE Spar Positions	
Structural And Aerodynamic Variables	Thickness/Chord	
	Chord Lenghts	Fuselage Length
	Spans	Fuselage Crossection
	Wing Streamwise Position	Nose Length
	Wing Vertical Position	Rear Length
	Tail Vertical Position	
	Sweep Angles	
Aerodynamic Variables	Dihedral Angles	
	Twist Angles	
	Nacelle Design	
	Airfoil Profiles	

Table 1 - Design variables

Constraints

The design space is limited by applying the main airworthiness requirements that can be properly evaluated with the adopted level of fidelity for the discipline simulations.

- Mission range = 2875 miles
- Cruise altitude = (15000,41000) ft
- Stress (tensile, compression, Von Mises) < stress_yield / 1.5
- Strain (wing) < 5%
- Angle of attack < 15 deg
- Elevator angle < 30deg
- Aeroelastic Damping > 2%
- Take Off distance < 7700ft
- Landing distance < 6200ft
- Missed Approach Climb Gradient > 2.1%
- Second Segment Climb Gradient > 2.4%

3. Integrated environment and multidisciplinary analysis

Analyses performed

A subset of load cases for the conceptual design optimization is selected, in order to verify the main requirements of FAR-25 (Federal Aviation Regulations Part 25 – Airworthiness Standards for Transport Category Aircraft).

In detail, the following structural and aeroelastic high-fidelity analyses are performed:

- WEIGHT ESTIMATION: the primary structure weights of wing box, fuselage and tail box are evaluated for the current choice of geometry and structural properties;
- STATIC TAXI BUMP: a 2g gravity load is applied at landing gear grid points;
- STATIC AEROELASTIC MANEUVERS: The longitudinal trim aeroelastic analyses for the following load factors are performed in order to simulate real load condition due to maneuvers from the flight envelope:
  - 2.5g full payload and full fuel
  - 2.5g full payload and zero fuel
  - -1.0g full payload and full fuel
  - -1.0g full payload and zero fuel
- FLUTTER: the dynamic aeroelastic stability analysis is performed at 1.2V_d. The aero-elastic damping coefficient is verified to be more than 2%.

In addition to the previous structural and aeroelastic analyses, the mission is analyzed. The most critical conditions are simulated, such as the landing and the take-off with one inoperative engine. In details:

- AERODYNAMIC ANALYSIS: the aerodynamic performance of the current design is evaluated on the basis of empirical flight test data corrected by high-fidelity simulations. The aerodynamic matrix is tuned by modifying specific factors, which allow to take into account the concept peculiarities and to better estimate the profile drag, the zero-lift drag and the friction drag. The aerodynamic coefficients and the polar curve are evaluated.
- MISSION PROFILE: the entire mission cycle (take-off, cruise, landing) is calculated for the required range and for the used propulsion system. The optimal altitude for the current design is evaluated and used for the cruise segment. All aerodynamics and mass properties are used for this analysis, and moreover the mass breakdown due to the fuel burn is taken into account during the mission calculation.
- TAKE-OFF AND LANDING: the takeoff and landing module computes the all-engine takeoff field length, the balanced field length including one-engine-out takeoff and aborted takeoff, and the landing field length. The approach speed is also calculated, and the second segment climb gradient and the missed approach climb gradient criteria are evaluated.
- FUEL BURN: the total fuel request for the entire mission cycle (take-off, climb, cruise, landing) is evaluated for the current design
- DRAG ESTIMATION: the drag is estimated during the entire mission cycle, taking into account the variation of the fuel weight during the mission. The drag estimation will be improved by an high-fidelity aerodynamic solver.
- SUBSYSTEM SIZING: all of the subsystems and control systems are sized for the current design. Therefore, on the basis of the actual structural weight evaluated by high-fidelity estimation, the total Gross Weight of the current design is calculated.

Disciplinary tools

In order to perform such analyses, specific tools disciplinary tools and solvers are needed.

- FEMWING: it is an aero-structural tool for the MDO of aircraft developed within this research activity at Mechanical and Aeronautics Dept. of Sapienza University of Rome. It can accomplish the following tasks: i) Parametric mesh creation, ii) MSC.Nastran coupling for high-fidelity static, dynamic, aeroelstic analyses, iii) Postprocessing of results, iv) Able to be coupled with any optimization code. It carries out all the structural and aeroelastic analyses listed in the previous section.
  The aerodynamic and structural Finite Element Models are generated for each design variable combination during the optimization.
  Figure 2 Example of structural and aerodynamic mesh built by FEMWING for MSC.Nastran solversshows an example of structural and aerodynamic models created by FEMWING for a OWN vehicle geometry.
- FLOPS: FLight OPtimization System (FLOPS) is an aircraft performance and gradient based optimization program by NASA for the preliminary and conceptual design, which is originally based on empirical and analytical models. It has been developed by including more advanced models and the possibility to analyze unconventional configurations. All the previous mission simulations, together with the performance evaluation and subsystems sizing, are carried out by FLOPS.
- Cart3D: is a high-fidelity inviscid analysis package (Adjoint-Based Aerodynamic Shape Optimization). It has been used for the aerodynamic matrix tuning in FLOPS.
- ModeFRONTIER: It is a the process integration and optimization software used for the implementation of the multidisciplinary analysis and the optimization.

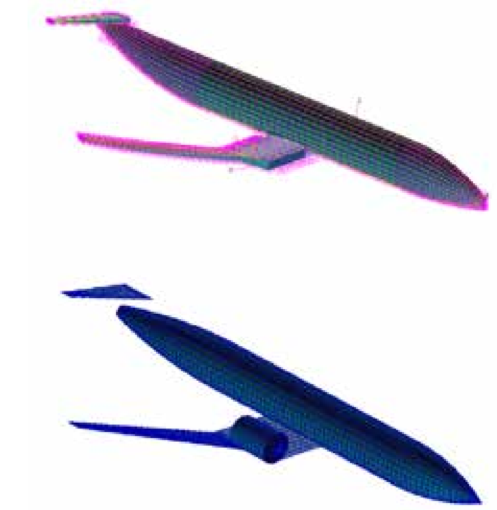


Figure 1 - Example of structural and aerodynamic mesh built by FEMWING for MSCNastran solvers

Multidisciplinary analysis (MDA)

Involving multiple specific evaluators allows to improve the fidelity of the multidisciplinary analysis, because the simulation of the disciplinary behaviors of a design can be carried out through the most appropriate method. On the other hand, it makes the complexity of the problem grow, because the coexistence of multiple function evaluators requires that the architecture of the optimization process guarantees the consistence of all of the disciplinary simulations with respect to the current design.

The approach used in this application aims to guarantee the consistency of the MDA at each iteration, by implementing a systematic exchange of coupling variables within the same function evaluation, without introducing variable copies and consistency constraints. In fact, as the parallelization of the function evaluation makes the process easier to be implemented, it may increase the number of designs evaluated before the convergence is reached. Therefore, when, as in the present case, the function evaluation is quite expensive, it is worth to spend a bit longer time in order to implement a monolithic Multidisciplinary Feasible Architecture. The MDO is carried out by combining all the discipline analyses, in order to sequentially evaluate the coupling variables necessary for the subsequent analyses. Figure 3 shows the MDO workflow, implemented in modeFRONTIER environment.

The four main function evaluation modules are executed in the following order:

- TAIL SIZING: since this first optimization is focused on the wing design and in order to reduce the number of design variables, a regression model, able to provide a reasonable sizing of vertical and horizontal tail, for the current dimension of fuselage and wing, is used.
- REDUCED FEMWING: It evaluates the primary structural weights of wing, fuselage, tail through an high fidelity calculation. It takes also into account joints and bolts by using a material density correction.
- FLOPS: it performs the mission analyses by using the actual weight, consistent with the current design. FLOPS evaluates the mission and aerodynamic performance, the necessary fuel together with the sizing of the subsystems.
- FEMWING: Once all the non-structural masses are known, FEMWING builds the complete structural FE-Model, comprising of subsystems, payload, fuel weight and performs the structural and aeroelastic analyses of the current design.

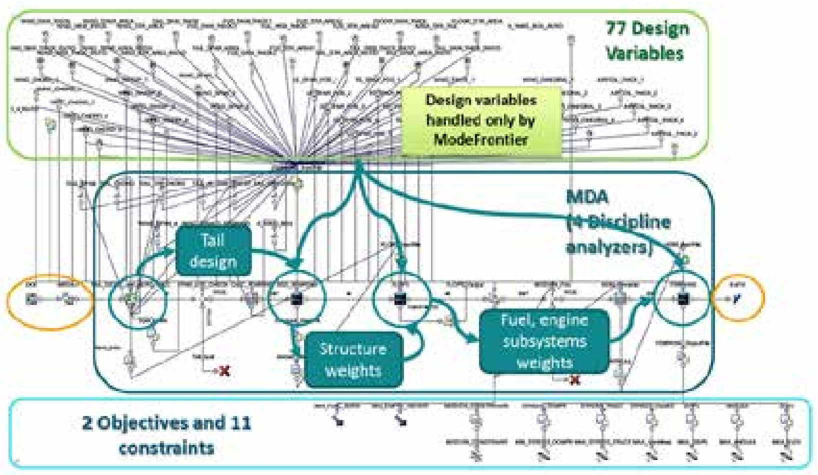


Figure 2 - Computational environment: modeFRONTIER user interface

Optimization steps

The optimization analysis is carried out by following three main steps (see Figure 3).

1. Statistical analysis of the design space (DOE)

The Statistical Analysis is used to better initialize the algorithm (find most signifcant input variables, proper side constraints, efficient DOE as initial condition of the optimization). In this application it was important to verify whether the MDA can capture the overall behavior we were interested in. For instance, It was important to check that the fuel weight has an inverse non-linear trend with the L/D, which confirms that the simplified aerodynamic model used is sufficient for this purpose. Thus, the FW is an indirect measure of the aerodynamic performance. The statistical analysis is used also for evaluating the correlation among candidate objectives and their sensitivity to the different disciplinary behaviors. As a final result, the initial DOE for the next optimization analyses is determined and the choice of the objectives is taken.

2. Structural SOO (TOGW minimization)

The baseline of the OWN does not comprise an airframe design, because it has been optimized only from the aerodynamic and engine points of view. Therefore, a first optimization process, in order to obtain a good starting guess of the airframe, will be done. In detail, the first structural optimization will be performed by using only the structural variables, while the external geometry will be frozen to Andy Hahn's baseline design. Genetic Algorithm (MOGA II) has been applied for this first optimization, because there is no initial data and a wide exploration of the design space should be done. In this phase the TOGW is used as only objective. In order to reduce the design space yet maintain diversity for the next optimization, the new side constraints for the structural design variables have been identified via clustering a wide set of suboptimal designs. As a result, the structural design variable side constraints to be used for the next optimization steps are determined.

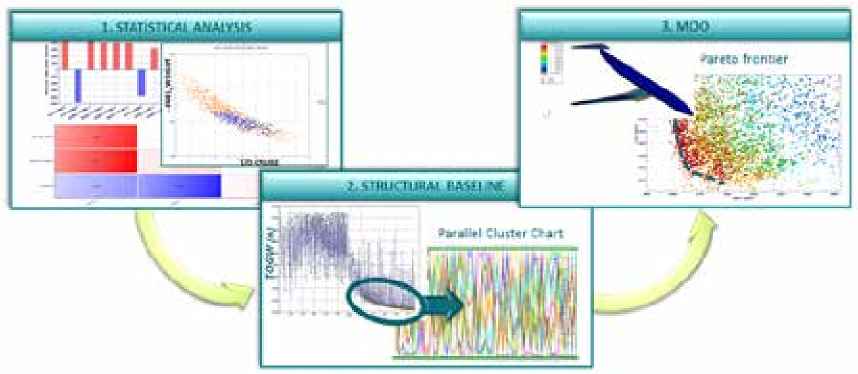


Figure 3 - Optimization steps

3. Multidisciplinary MOO for the OWN

Once an initial guess of an acceptable airframe is obtained, the multidisciplinary optimization of the whole model can be performed by using all the design variables, both aerodynamic and structural variables, whose ranges are centered on their reference values.

This multi-objective optimization problem is intended to solve explicitly the minimization of the two objective functions EW and FW, subject to the constraints listed before.

The MOGA II genetic algorithm is employed for the optimization.

Once a set of sub-optimum designs is obtained and the flutter check starts failing, the flutter analysis is introduced into the MDA and it is systematically performed for the evaluation of all of the design samples. In this second optimization phase, the population size is gradually reduced.

In order to lead toward the real Pareto frontier the optimization and speed up the convergence, an interactive approach is applied. Indeed, the genetic algorithm is able to converge to the global optimum solutions, but the convergence speed may be quite low when the asymptotic trend is reached. The interaction between user and optimization process is based on four principles: multi-step optimization, population size reduction over the optimization, multi-fidelity approach, and design interchange among different optimization analyses running in parallel. The most expensive analyses are included subsequently when sub-optimal designs have been obtained.

4. Pareto frontier and significant Pareto designs

Figure 5 shows three significant designs from the Pareto frontier, having respectively the minimum value of EW, TOGW and FW. The structural thickness is represented by a color scale, while the lifting mid-surface represented by the grey surface.

Indeed, the minimum FW design has the highest value of the aspect ratio and of the total span, while its root chord of the outboard section is the lowest one. Its mean thickness-to-chord ratio is the lowest one. As a consequence, in order to guarantee a proper bending stiffness, the spar caps and webs increase their area and thickness and the wing box (the spar positions) moves forward. The minimum EW design has the opposite trends, while the minimum TOGW design is always a trade-off between the other two.

As previously hypothesized, the minimum EW design and the minimum FW design represent the extreme solutions of the Pareto frontier, and are located respectively on the top and on the bottom of the frontier. The minimum TOGW design is one of the compromises between them.

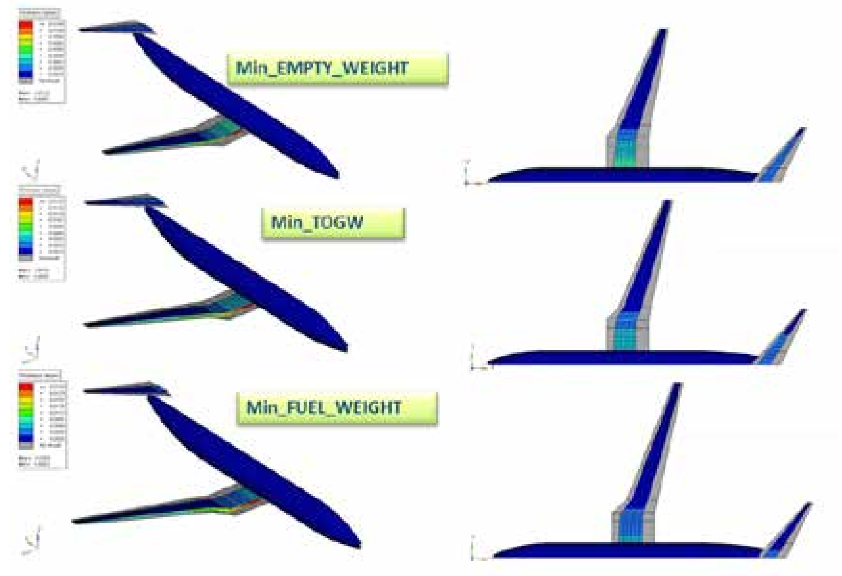


Figure 4 - Significant Pareto designs, isometric and plant view

5. Concluding remarks

The advantage of the multi-objective approach is presented in Figure 5 and Figure 6, where the designs are classified on the basis of, respectively, their TOGW and wing Aspect Ratio through a color scale. The result of the MOO is the Pareto frontier, that is identified by the red line.

The optimal designs sweep a wide range of configurations. In fact, as shown in Figure 7, The aspect ratio of the Pareto designs sweeps between around 12.5 and 9.85, with a total variation of 26%, while the L/D of the minimum FW design is 9.41% higher than the minimum EW design one. In addition, it is interesting to notice that the FW can be at maximum decreased by the 6.24% against a EW increase by 3.68%. Whereas, the TOGW range is very small, namely, its variation among Pareto designs is around 0.68%.

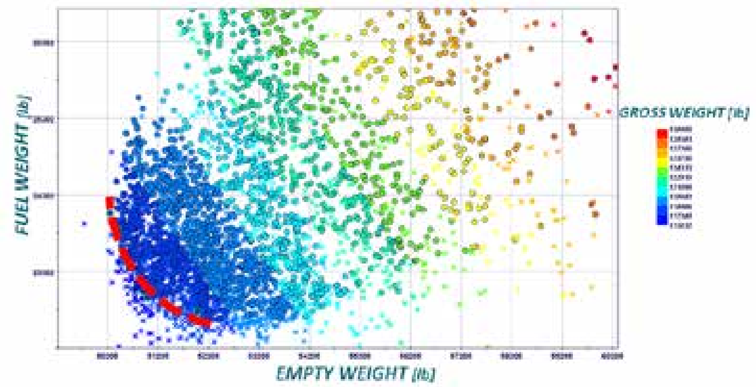


Figure 5 - Pareto frontier in the objective space of Empty Weight and Fuel Weight, vs. Take-Off Gross Weight

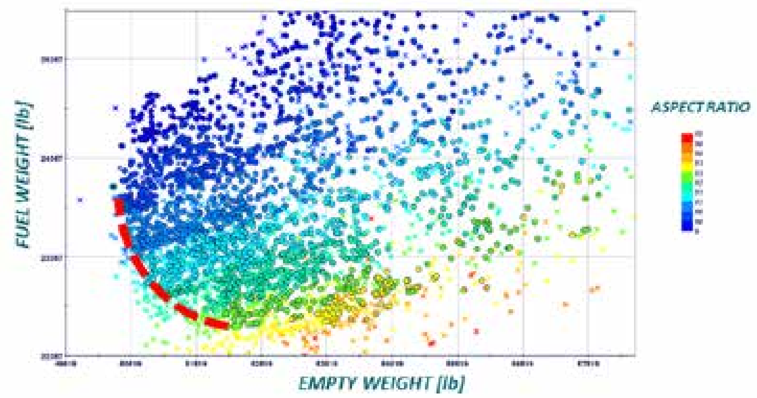


Figure 6 - Pareto frontier in the objective space of Empty Weight and Fuel Weight, vs. Aspect Ratio

Therefore, the obtained designs are all different compromises between structural and aerodynamic requirements for a quasi-constant value of TOGW.

Optimizing the two components of the TOGW has provided the designers with a rich design set, from which it is possible to get useful information about the system on hands. Following a A posteriori approach, not all designer preferences are included in this first phase. Further preferences and requirements can be applied for selecting the candidate designs, on the base of the results. An open-mind approach and human final choice may provide better (and newer) results.

Moreover, the proposed MDO-MOO framework, along with the optimization strategies, allowed to reduce computational time of stochastic global algorithms.

Image at the top of the article: The Over-the-Wing-Nacelle concept

Stefania Gemma
Aerospace Engineer at Airworks s.r.l. (Rome) and researcher at Sapienza University of Rome
(Dept. of Mechanical and Aeronautical Engineering)

For more information on Optimization Techniques:
Francesco Franchini, EnginSoft
f.franchini@enginsoft.it

Optimization of Kaplan turbine through on-cloud simulation: The Fortissimo SuRE_HPC Project

Everyone is well used to the idea of cloud: nowadays more and more applications from PC to smartphones are based on cloud storage. Normal web activities like reading newspapers, looking for hotels, surfing on social networks operate through servers placed thousands of kilometres far from the end user.

Why do we not use the cloudification also for CAE simulations? Through the exploitation of high speed internet connections there is no need to buy in-house expensive computational clusters.

Fortissimo is a collaborative EU project with the precise purpose to enable European SMEs to be more competitive globally through the use of simulations services running on High-Performance-Computing (HPC) Cloud-based infrastructure.

The Fortissimo project (SuRE _ HPC, Sustainable and Renewable Energy) presented in this article is a clear example of optimization through on-cloud simulations: Zeco, an Italian SME in the hydro power sector, wants to develop a new optimal concept of turbine in half of the usual time. Hence a customized tool for water turbines design, based on Computational Fluid Dynamics (CFD) simulations on HPC cloud infrastructure, is developed.

Thanks to the strong collaboration with EnginSoft, the CFD expert, and CINECA, the HPC provider, Zeco fully achieves its goals.

Industrial Setting

There are many companies involved in the renewable energy sector in Europe. Many of these are small and medium-sized enterprises (SMEs). There is significant political and industrial demand for the development of new efficient systems which are able to increase the exploitation of renewable energy resources: solar, wind and hydro power. Among these, water turbines are the oldest renewable energy system, as they have been employed since XIX century. Nowadays this sector is gaining more and more interest because it represents at the same time an efficient, reliable and low-impact energy system. The focus of the activity is on small plants (net power produced between 100 kW and 10 MW), which are able to exploit a larger part of the natural hydro power potential with a very low environmental impact. Kaplan water turbines (Figure 1) are widely used because of their easy adaption to different installation conditions, preserving high efficiency. Indeed the possibility to regulate both stator and rotor blades makes the Kaplan turbines able to keep high performances even in off-design conditions, hence they are able to work efficiently in a wider range of operating conditions compared to other turbine types (Francis, Pelton). However hydro power plants usually require significant civil works and costs that can be a serious obstacle to small and micro installations. Kaplan turbines for example are usually fed by radial inlet systems which are much more expensive than axial feed ones. The development of such systems can be now part of a dynamic and profitable market. They can reduce installation costs increasing the exploitation of hydro power resources at a lower cost. One of the most important problem is to adapt the axial Kaplan turbine system at different installation conditions, preserving the high efficiency and avoiding the occurrence of critical issues, like cavitation.

How CFD can get a better Water Turbine Design?

Computational Fluid Dynamics (CFD) simulation can dramatically improve the turbine design process and reduce the number of trials needed to test the final product. These simulations, however, are CPU-intensive and require the use of HPC and need the support of domain specific expertise. Nevertheless, their use can improve competitiveness and reduce the 'time-to-market' for new products, whilst increasing the reliability.

In addition the use of physical complex models, like the multi-phase approach, and the large dimensions of the area simulated, makes the HPC mandatory to simulate in a short time such a plant.

As a matter of fact, the traditional trial-and-error approach of European SMEs in the design of hydro power turbines does not easily support the characterization of cavitation and transient phenomena, which is more commonly an approach used by premium global manufacturers. This reduces the competitiveness of European SMEs in the global market. CFD simulation codes are useful tools for the design of water turbines particularly when highly challenging aspects like cavitation have to be taken into account. However, in many cases, such codes are too complicated to be used by SMEs with limited resources and require access to supercomputers. This project brings together the supercomputing resources, domain specific expertise and licensed simulation packages which can overcome these barriers. This enables European SMEs like Zeco to compete globally through better design and quicker times to market.

Figure 1 – Kaplan (left) Francis (top) and Pelton (bottom) turbines

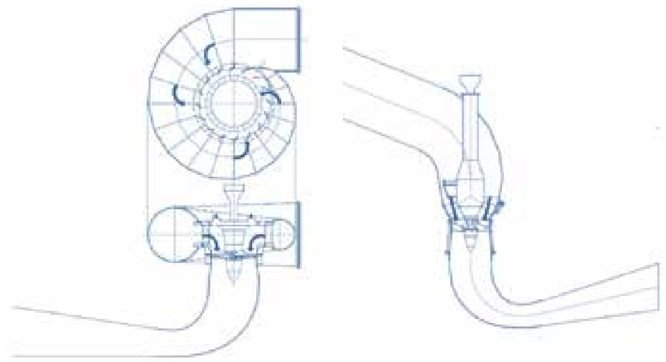


Figure 2 – Standard radial inlet layout (left) and axial inlet layout (right)

Virtual Customization Tool Development

The featured CFD analysis has the goal to determine the Kaplan turbine's geometry for a new axial distributor layout, in order to maximize the efficiency at a constant rotational speed. The final aim is the turbine's hill chart, or performance mapping at different operating conditions.

To achieve this, the complete analysis requires the following steps:

1. Geometry parameterization
2. Parameter screening at nominal water flow rate
3. Efficiency charts at different water flow rates
4. Turbine hill chart

As shown in Figure 2, the axial layout geometry differs from the original radial one in the rotor feed system (distribution channel and stator blades). Instead the rotor and diffuser parts are not modified. Compared with the original radial layout, the new axial layout is cheaper and simpler, since flow distribution is made through a tube and not through a spiral volute, increasing the turbine versatility.

Such versatility can be obtained with fine tuning of all the characteristics of the machine, i.e. tube length, correct blade inclinations, etc. For this reason, in the virtual tool, all the coloured parts in Figure 3 and Figure 4 are suitable for parameterization: for what concerns the Kaplan rotor and stator blades (blue and red part respectively) there's only one degree of freedom which is the angle of rotation with respect to each blade axis and then the objective is to find the correct blade inclination at the given operating conditions that maximizes the net rotor work (or efficiency).

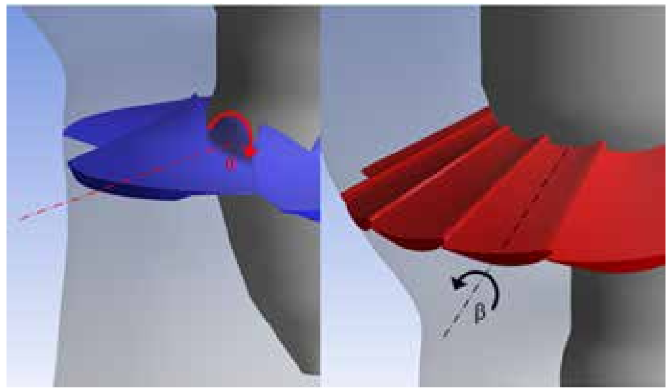


Figure 3 – Rotor (left) and stator (right) blades inclinations

For what concerns the fixed distribution blades (green part), the main variable is the blades displacement respect to the mid inlet plane (shown in violet in Figure 4). Also the distribution channel geometry (shown in light blue Figure 4) has to be optimized, as the inlet flow angle and the height with respect to rotor blades can influence turbine performances.

All other geometry parts are considered as constraints in the optimization process. Also part of the operating conditions, such as the rotational speed for example, are considered as constraints.

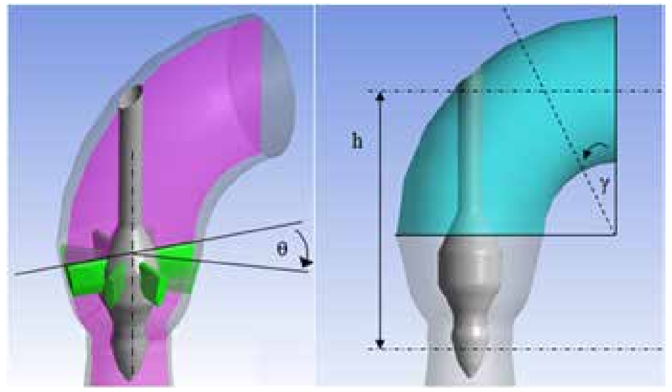


Figure 4 – Distributor blades angular shift (left), distributor channel height and inclination (right)

Following the parameters and constraints identification, the optimization setting process involves the choice of the expected ranges of values which reasonably limit the parameters. This is a very important step as it defines the extent of the Design of Experiment matrix (DOE): the more accurate is the choice of the parameters limits, the more the matrix will be dense around the potential optimum, improving accuracy and reducing simulation time.
Of course the correct identification of the output parameters is also fundamental.
The main output in a water turbine is the efficiency, which is calculated as the ratio between the net head produced by the turbine and the gross head. The efficiency is normalized to the original radial layout value. Objective of the optimization is to maximize the turbine efficiency (η):

$$\eta = \frac{\frac{C \cdot \omega}{\dot{m} \cdot g \cdot H}}{\eta_{ref}}$$

where C is the torque produced by the turbine blades, ω the rotational speed, $\dot{m}$ the water mass flow rate, g is the acceleration due to gravity, H is the maximum allowed vertical distance between the upstream's surface water forebay elevation and the downstream's surface water elevation at the tailrace and η_{ref} the reference efficiency.

Figure 5 – Cavitation effects on turbine rotors

Finally the last output parameter is a control variable: a common issue of water turbo-machinery is cavitation, or the formation of water vapour bubbles in zones (usually in rotor blades proximity) where static pressure may fall below the vapour pressure value. It is very dangerous because it causes a drop of turbine performance and erosion of blades (Figure 5). In order to take account of this phenomenon, a CFD multiphase model is employed, and cavitation is monitored as the percentage of rotor fluid volume actually occupied by water vapour.

For the model pre-processing, the aim is to create a highly automatic interface which allows geometry modification and mesh generation only by setting particular values of input parameters, within the limits fixed in the previous steps.
ANSYS Workbench is the chosen suite, as it can manage all the project phases, from geometry creation to final optimization and post-processing, in one single visual interface. Figure 6 shows the project schematic as it appears in the ANSYS Workbench interface (Release 16.2).

ANSYS CFX is the CFD software tool used in the project due to its remarkable accuracy, robustness and speed with rotating machinery such as hydraulic turbines.
The real plant exploits the difference of height between two water basins, upstream and downstream, together with the river flow rate fraction that is driven to the turbine (Figure 7). This flow generates a torque on the turbine rotor, which is converted in electricity. The CFX model doesn't reproduce explicitly the water basins, but it simulates their presence imposing the ambient pressure at the outlet, the mass flow rate at the inlet, and the effect of gravity on the system.

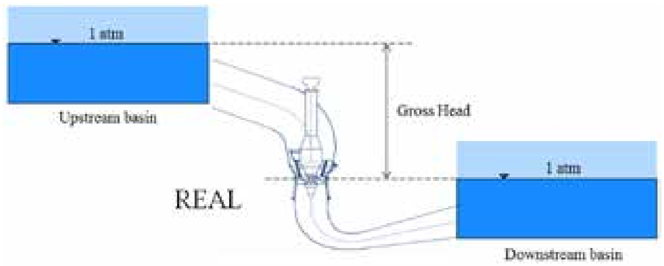


Figure 7 – Real plant concept

Once the Workbench project is completed and functional in all its steps the design exploration process can start. First, the type of Design of Experiments (DOE) has to be set and it depends on:

- Input parameters (5): α, β, γ, h, θ
- Output parameters (3): H, η, cav%
- Cloud HPC availability: 128 cores from CINECA Galileo cluster

Figure 9 shows the hexahedral mesh of the stator part of four different design points, created automatically in TurboGrid during the Workbench project update.
Each design point is simulated (on cloud) with a steady state analysis running in parallel on 128 cores of the Galileo cluster. Geometry and mesh pre-processing, together with CFX setup and post-processing, are run in serial. Table 1 lists the average running time for each process.

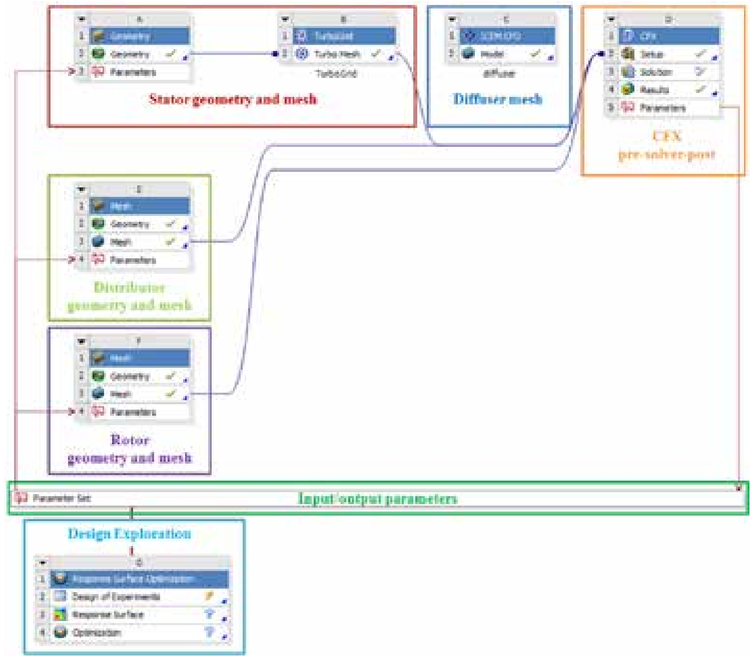


Figure 6 – Workbench project schematic

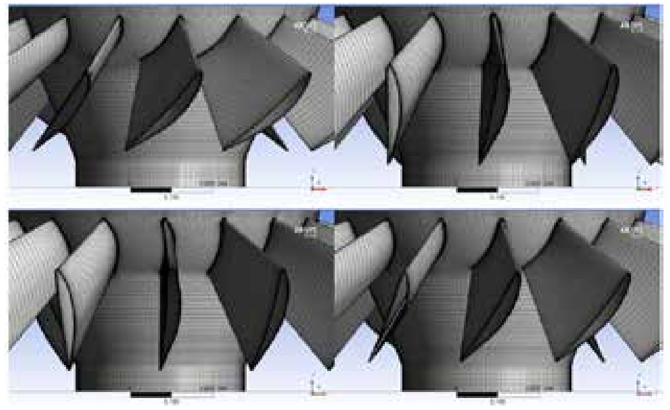


Figure 8 – Stator mesh at four different β values

Process	Cloud		Local	
	Cores	Time [h]	Cores	Time [h]
Geometry and mesh pre-processing	Serial	1	Serial	1
CFX setup and post-processing	Serial	0.25	Serial	0.25
CFX steady state multi-phase analysis	128	1.5	8	20

Table 1 – Process average timing list on Galileo cluster

Each design point has a different combination of geometries, or meshes, while the water flow rate is kept at this round constant at the nominal value (100%).
Figure 9 shows DOE four samples, with velocity streamlines in the rotor sections.

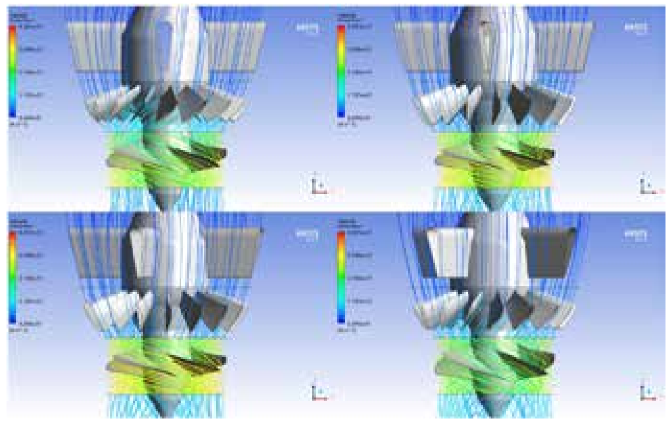


Figure 9 – Velocity streamlines in four different design points

Aim of this first round of DOE analyses is to find parameters correlation and influence on turbine performance in terms of gross head and efficiency. Through design exploration, a response surface is created, which allows to understand which input variable has a really important impact on efficiency and which has not: in this case alpha is the first parameter in terms of importance, while beta is the second. The other variables influence on outputs is not significant.
Hill chart is the performance map specific of the turbine: it relates plant operating conditions, gross head and water flow rate, directly to the output efficiency. It is very important as turbine business case considerations are based on hill chart data.

Hill chart creation needs the turbine geometry to be optimized at different flow rate values. As a matter of fact, water flow rate is no more a constraint but fully an input variable.
The parameters reduction (from five to two) achieved in the first screening phase simplify considerably the optimization process, making next steps easier. New parameter set is reported in Table 2.
Turbine hill chart is derived from response surfaces at four different water flow rates: 100% (nominal flow rate), 125%, 75% and 50%. The optimization process employed for the first (nominal) case is the same than the other mass flow rates.
As the design space is much more concentrated than before, the Central Composite Design (CCD) is the used DOE type. Every design point simulated is a possible real working point of the turbine.
The resulting response surface is showed in Figure 10, with η as a function of α and β.
The algorithm used is DesignXplorer genetic aggregation that guarantees similar results compared to standard 2nd order polynomial surface with a higher goodness of fit ($R2 = 99\%$).

Type	Parameter name	Symbol
INPUT	Rotor angle	α
	Stator angle	β
	Water flow rate	Q
OUTPUT	Gross head	H
	Efficiency	η
	Cavitating volume	cav

Table 2 – Hill chart parameters list

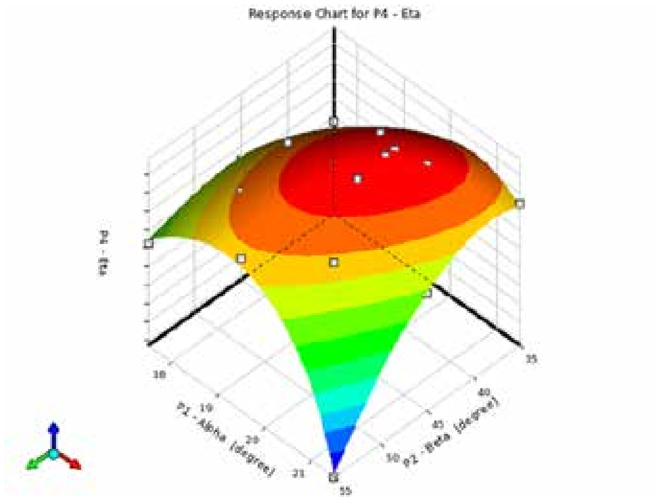


Figure 10 – Response surface, 100% water flow rate case

Once the surface response is done, the optimization process can start: there are many combinations of input parameters (α and β) that guarantee a particular value of gross head (and water flow rate), but only one couple has the best efficiency. The solution is to screen the response surface maximizing the efficiency.

Figure 11 plots all the virtual points generated by screening optimization, as $\eta = f(H)$. The border on the point cloud top is the Pareto front, or the ensemble of couples α-β that guarantees the maximum efficiency for each gross head value. The Pareto front is the first one of the four input curves of the final hill chart.

Regarding cavitation, Figure 12 reports an example where the increasing gross head operating conditions makes the cavitation becoming more and more relevant. In order to keep cavitation within very strict limits (less than 0.5%), the maximum gross head

elaborated by the turbine at nominal water flow rate should be smaller than 29 m.
The same approach followed for 100% water flow rate characterization is repeated for the other three additional operating conditions.
The final hill chart response surface is obtained from previously calculated Pareto fronts, with a genetic aggregation algorithm interpolation. Figure 13 shows resulting 3D surface, with highlighted design point curves. The appropriateness of its fit is very good, higher than 99.5%.
Figure 14 shows the standard hill chart diagram, with contours of efficiency as a function of gross head (H) and water flow rate (Q). Top of the hill, or the maximum efficiency point, has a calculated efficiency very similar to the original radial layout one. The cavitation risk zone is located on the top-right part of the diagram. The chart meets Zeco expectations.

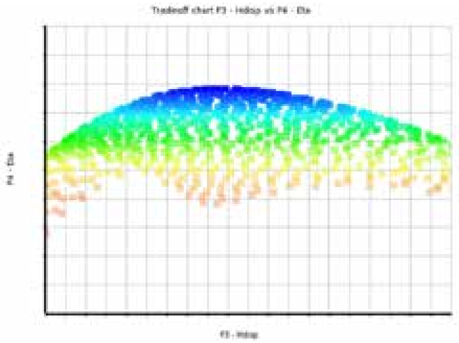


Figure 11 – Efficiency as gross head function, 100% water flow rate case

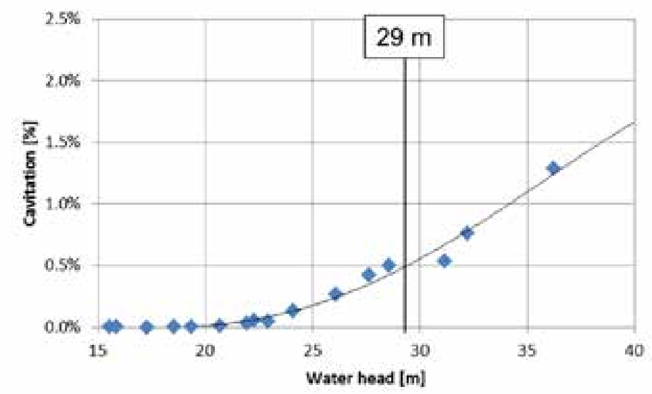


Figure 12 – Volume of cavitation as a function of gross head, 100% water flow rate case

CONCLUSIONS

The collaboration among the three partners, Zeco, EnginSoft and CINECA, produced within the Fortissimo framework for the development of a highly automatic customized tool for water turbine design through cloud based technology.
From a technical point of view this tool developed by EnginSoft enables Zeco to understand performances and issues of its product in a new market framework. For business considerations it is essential for Zeco to identify the performance over a wide range of scenarios (hill charts) of its product (hydraulic turbine) in order to quantify and optimize its proper position on the competitive market. Indeed the turbine performance map is the only and most important data needed to decide if the project can be industrialized and sold with a competitive edge or not.
The use of HPC resources supplied by CINECA led Zeco to characterize the turbine hill chart map within a very short time compared to in-house practice. As a matter of fact the time required for all the computations made in this experiment in a standard workstation (8 cores) can be estimated in about 1 year, while with the HPC solutions all the simulations took 3 months, with around 75% of time reduction. It should be noticed that such a reduction makes the difference whether customization through simulation can be done or not.

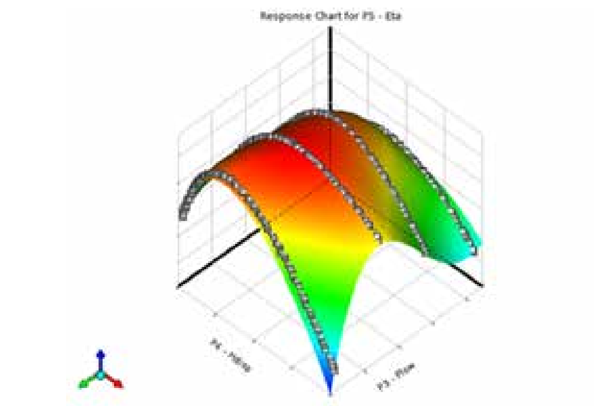


Figure 13 – Hill chart response surface

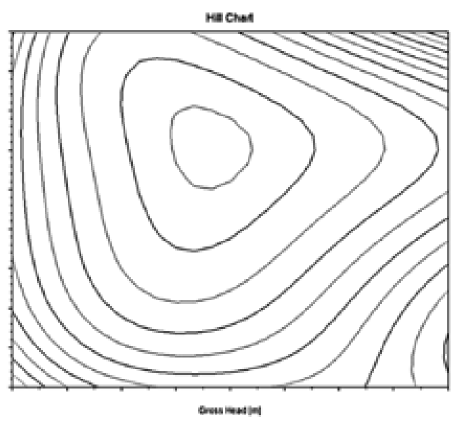


Figure 14 – Hill chart top view, relative efficiency as a function of gross head and water flow rate

Figure 16 – Partners involved in Fortissimo WP519

Acknowledgments
The research leading to these results has received funding from the European Union Seventh Framework Programme ([FP7/2007- 2013]) under grant agreement n° 609029.
For further details, please visit FORTISSIMO's website: www.fortissimo-project.eu

Riccardo Bergamin, R&D Manager
Francesco De Pretto, Technical Manager
ZECO di Zerbaro & Costa e C. Srl

Claudio Arlandini, HPC Project Manager
Ivan Spisso, HPC CFD Expert
CINECA

Lorenzo Bucchieri,
Michele Andreoli,
Alessandro Arcidiacono
EnginSoft Italy
l.bucchieri@enginsoft.it

Debris flow simulation on the debris flow breaker

Japan is a long archipelago stretching from north to south, which has a variety of geography, geological conditions, and climate in its narrow land area, we can enjoy the beautiful change of seasons in each site. However, at the same time, Japan is under the severe natural condition which easily produces natural disasters including earthquakes, heavy rain, heavy snow, landslides, and debris flow, that is very unique in the world. KITAC CORPORATION, a general construction consulting firm, has a very important role to provide services such as geological research, design management of civil engineering structures, environmental analysis, and disaster recovery, in such a country where various disasters often occur, to support convenient, easy and safe human life. In this article, the evaluation example about the advantages of the debris flow breaker which is used to reduce the damage produced by the debris flow, using MPS, the Moving Particle Simulation method.

There are some sediment disasters every year in Japan. Especially the disaster in Hiroshima caused by heavy rain in August 2014 inflicted terrible damage, which was the worst in the last 30 years. At that time, the series of massive debris flows occurred because of local heavy rain for a short time and the collateral landslides behind the residential areas. Such debris flows which happen in overpopulated districts cause severe damage to residential areas and any protection system is urgently required. Concerning the reduction of the landslide disaster damage, one of the countermeasures is to install a debris flow breaker. A debris flow breaker is a gridiron-shaped screen placed in a riverbed and it has the ability to separate the solid and liquid from the debris flow, which can reduce tractive force. A real debris flow breaker is shown in Fig. 1. KITAC evaluated different debris flows traveling down on the debris flow breaker, to find the optimal specs for the debris flow breaker. As debris flow is a two-phase flow of solid and liquid, and the interaction between gravel and muddy water, collision of gravel and extreme change of free surface should be modeled in the simulation. However, it's not easy for conventional grid method CFD software tools to simulate such problems especially for complicated free surface problems. Therefore, they choose MPS which shows accurate results qualitatively for incompressible fluid analysis and can stably solve dynamic free surface problems, to simulate the debris flows on the debris flow breaker. The software product that they used is Particleworks using MPS, developed by Prometech Software.

For creating the simulation model, muddy water of debris flow which is the liquid phase was assumed as non-Newtonian fluid (Bingham fluid) which viscosity is depending on the shear velocity. As a Bingham fluid model, Particleworks employs a bi-viscosity model that treats a fluid as a viscoplastic fluid in a fluid state, and as a highly-viscous fluid in a rigid state. Gravel of debris flow which is the solid phase is exactly not rigid, but it can be considered as approximately rigid compared to the water

Fig.1 - Debris flow breaker

flow deformation resistance. Therefore, the gravel was assumed to be rigid and was modeled using particle cluster having the same mass as fluid. (Fig.2)

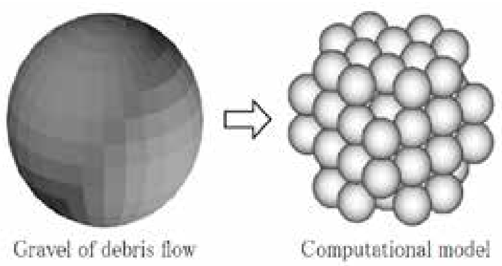


Fig.2 - Computational model for the gravel of debris flow

For the simulation model, a horizontal screen of 60cm long which space was changed between 0mm, 8mm, and 16mm was set on the rectangular flume of 350cm long, 20cm wide, and 17.6° gradient (Fig.3). The material which slip condition was given between flume and initially put on the riverbed as sediment, and the mixed muddy water and gravel were placed on top of it. Then, the debris flow was created by flowing muddy water from the upper stream. The influx was defined as 2 cases of 2.0l/s and 5.0l/s. The simulation was performed with approximately 300,000 particles, 5mm particle size, 10s simulation time, and 2E-4s time interval, for the limitation of the computer performance.

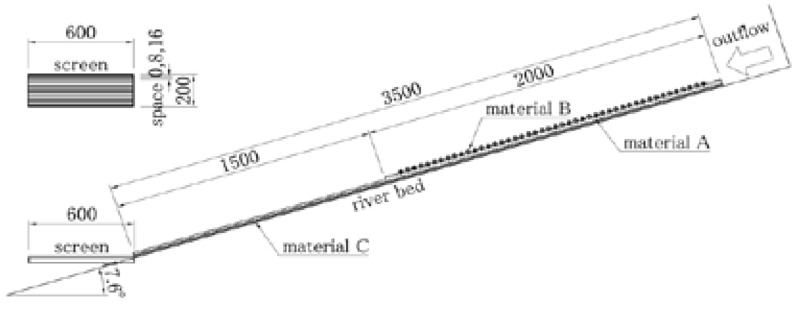


Fig.3 - The simulation model of the flume

Fig.4 and Fig.5 show the distribution of debris flow moving down on the screen in the cases of 5.0l/s influx, and 8mm and 16mm screen spaces. After the debris flow moving in on the screen, it separated into gravel and muddy water slowing rapidly. Finally debris flow breaker stopped the debris flow front. As a result, the muddy water sedimentation and run-up were confirmed because the gravel on the screen played a role of an obstacle. The large debris could be caught on the screen of 16mm space, while debris was pushed out from the edge of the screen of 8mm space.

Fig.6 shows the moving distance of the debris flow forefront and velocity of the gravel and muddy water. Here, the initial point of the moving distance is the upper edge of the stream. As the screen length is 60cm, the red dot line means that the debris flow went over from the downstream edge of the screen when the distance is over the screen length. One can see from Fig.6 left that there is no big difference in the moving distance of the debris flow forefront in the cases where the debris flow stopped on the screen, though the screen space was changed. Those simulation results were close to the previous experimental studies.

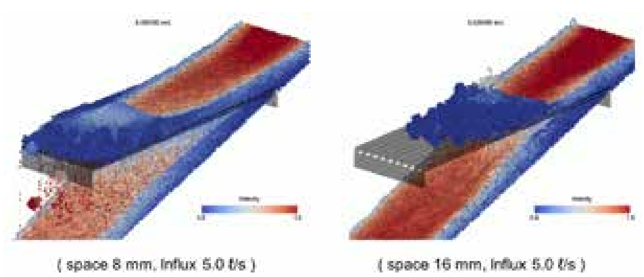


Fig.4 - The distribution of debris flow moving down on the screen (bird's-eye view)

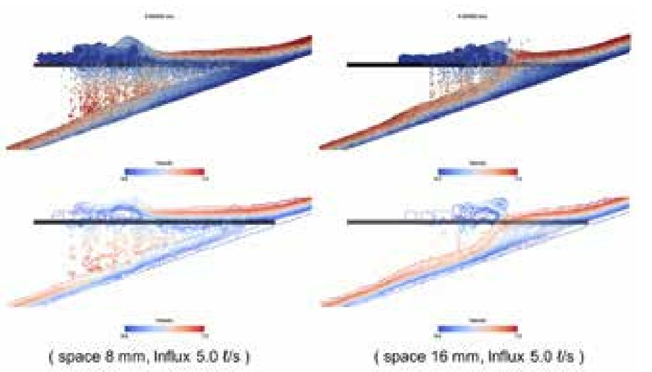


Fig.5 - The distribution of debris flow moving down on the screen (side view)

Conclusions

KITAC could simulate the debris flow on the debris flow breaker using Particleworks. We hope that such simulations would be one of the effective countermeasures against disasters which can happen everywhere. Further work is needed to improve simulation accuracy using additional material parameters based on field observation and particles of different size.

Acknowledgements

The author and Prometech Software wish to thank Mr. Hajime Ikeda of KITAC CORPORATION for the permission to have an opportunity to introduce the debris flow simulation on the debris flow breaker using Particleworks.

Reference

"Debris flow simulation on the debris flow breaker by using Particleworks" presented at Prometech Simulation Conference 2014.

Sunao Tokura, Prometech Software Inc.

For Particleworks inquiries:
Massimo Galbiati, EnginSoft
m.galbiati@enginsoft.it

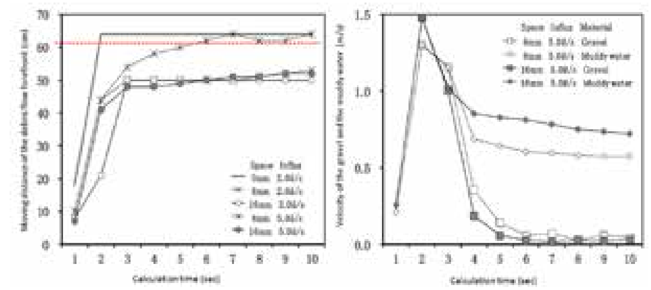


Fig.6 - Plot of calculation time vs the moving distance of the debris flow forefront (Fig.6 left) and, velocity of the gravel and muddy water (Fig.6 left)

Large Scale Structural Optimization with ANSYS Workbench

VR&D's newest offering is called GSAM (Genesis Structural Optimization for ANSYS Mechanical), structural optimizer GENESIS perfectly integrated in ANSYS Workbench. This product is an extension that adds multiple optimization capabilities like topology, topometry, topography, freeform to the ANSYS Environment, resulting of immediate use. The ease of use, however, must not be underestimated; the optimization "engine" is state of the art. GSAM utilizes the ANSYS thermo-structural analysis and the capability of GENESIS, a structural optimization program with proven experience in solving problems efficiently. This combination provides the analyst a powerful tool to lead towards the technical solution for the component with the best structural performance, in the early stages of the project, taking into account the production process and costs.

Vanderplaats Research & Development, Inc., (VR&D) was founded by Dr. Garret N. Vanderplaats in 1984 for the advancement of numerical optimization in industry. Over the years, the company has evolved into a premier software company, developing and marketing a number of design optimization tools. The company mission is to provide the best technology, software, experts and client support in the optimization world. VR&D's headquarter is located in Colorado Springs. They also have an office in Novi, Michigan.

VR&D's products can be divided in General Purpose Optimization Software (DOT, BIGDOT, VisualDOC) and Structural Analysis and Optimization Software (SMS, GENESIS, Design Studio, GTAM, GSAM, ESLDYNA).

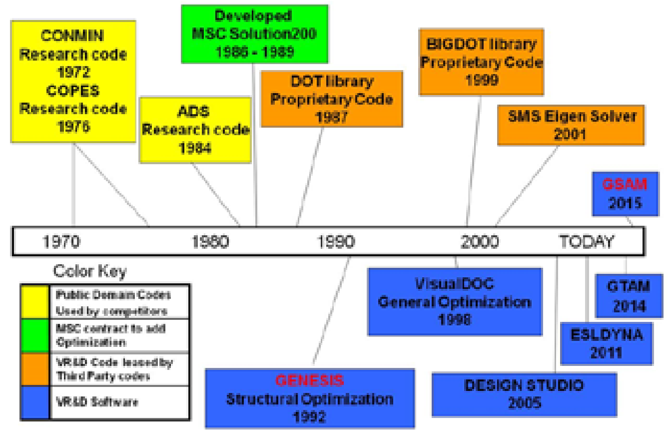


Fig. 1 - VR&D Product History

Design Optimization Tool DOT is a general purpose numerical optimization software library which can be used to solve a wide variety of nonlinear optimization problems. BIGDOT is intended to solve very large, nonlinear, constrained problems where gradient information is available, and function and gradient evaluation is efficient.

As mentioned previously GENESIS is the engine behind GSAM.

GENESIS is a fully integrated finite element analysis and design optimization software package.

Analysis is based on the finite element method for static, normal modes, direct and modal frequency analysis, random response analysis, heat transfer and system buckling calculations.

Design is based on the advanced approximation concepts approach to find an optimum design efficiently and reliably. An approximate problem, generated using analysis and sensitivity information, is used for the actual optimization, which is performed by the well-established DOT or BIGDOT optimizers. When the optimum of the approximate problem has been found, a new finite element analysis is performed and the process is repeated until the solution has converged to the optimum.

Many design capabilities are available: topology, shape, topography, freeform, topometry and sizing optimization.

GENESIS has fast and robust solvers for the finite element analysis part (SMS) and recently released state of the art optimizers (DOT,BIGDOT). That allows a quick convergence with less iterations for many types of problems.

GENESIS uses easy interfaces. Design Studio is a design oriented pre- and post-processor graphical interface for the GENESIS Software. It features built-in and easy-to-use trails for setting up the optimization problem and running GENESIS from the interface. It also supports post-processing of the optimization results with contour plots, deformed plots, animations, etc.

The other interface, in which VR&D has invested heavily for the last two years, is ANSYS workbench.

In 2014 VR&D released GTAM (GENESIS Topology for ANSYS Mechanical), an integrated extension that adds topology optimization to the ANSYS environment. Topology Optimization is an optimization

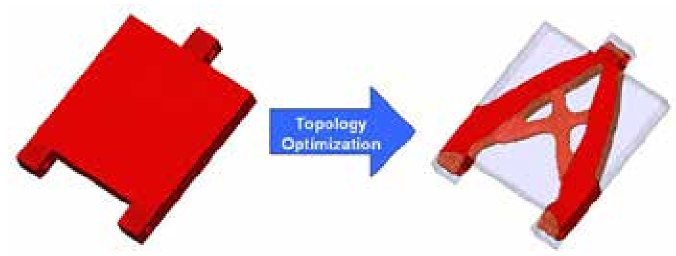


Fig. 3 - GTAM: Topology Optimization in ANSYS workbench

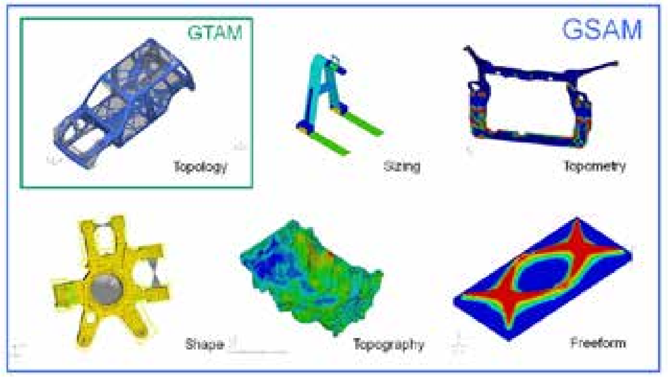


Fig. 3 - GSAM design capabilities

Following a brief description of GSAM capabilities to better understand differences and unique features.

Topology Optimization

Topology design is to find the optimal distribution of material in a given package space while improving the product performance. Typically topology optimization is used by engineers in the early design stages to generate innovative design proposals. GENESIS provides an extensive family of fabrication constraints such as casting, extrusion, symmetries, etc., that helps the user to obtain easy to build designs.

capability which allows the user to find the best distribution of material. With topology optimization, an optimal structure is generated by carving out material from a given design space, allowing for a given amount of material.

One year later VR&D released GSAM (GENESIS Structural Optimization for ANSYS Mechanical) an integrated extension that adds large scale Structural optimization to the ANSYS environment. GSAM can perform topology optimization (GTAM is a subset of GSAM) as well as topography, freeform, sizing and topometry design.

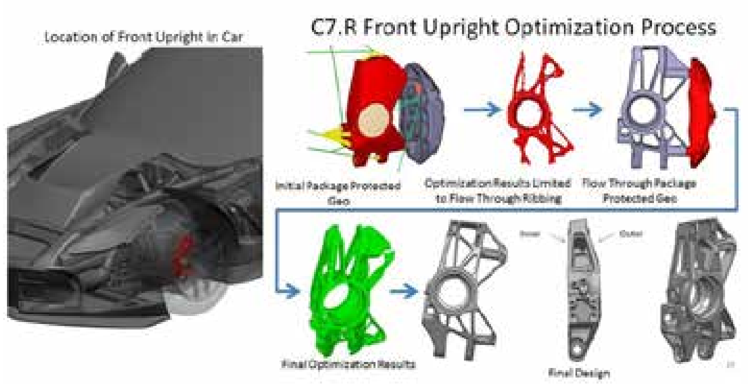


Fig. 5 - Tolopoly Optimization application (Courtesy of Pratt & Miller Engineering)

GSAM allows the integration of two powerful solvers, one is ANSYS for the analysis part and the other is GENESIS for structural optimization. ANSYS adds to linear, modal, inertia, heat transfer, buckling analysis, the possibility to perform also non linear, thermal- static and transient calculations. GENESIS adds structural optimization thanks to robust optimizers that speeds up the solving time, especially for large scale problem.
Designers benefit by automatically generating innovative designs in a reliable, robust and easy-to-use interface. GSAM helps to reduce cost and/or weight and to improve performances in the conceptual phase of a component design. The extension allows the user to setup the structural optimization problem, optimize, post-processing, export optimized geometry all within ANSYS environment.

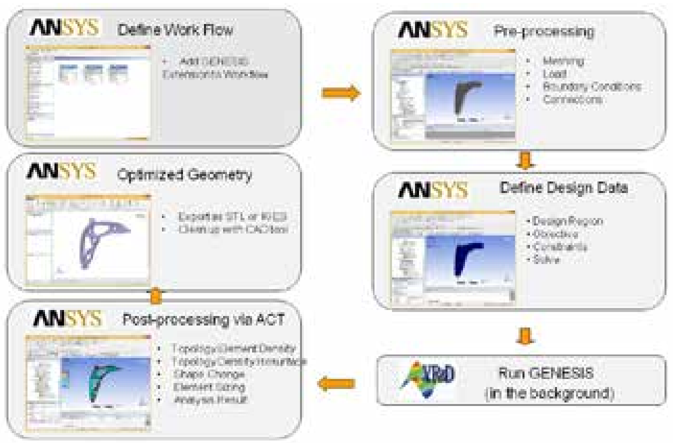


Fig. 4 - Combined Workflow ANSYS - GENESIS

Topography Optimization

Topography design is a special type of shape optimization where the grids are moved along the normal direction of selected surfaces. Typically topography optimization are used by designer to generate bead patterns on a plate to improve the design performance. Fabrication requirements such as symmetries, extrusion and bead fraction can be imposed if necessary.

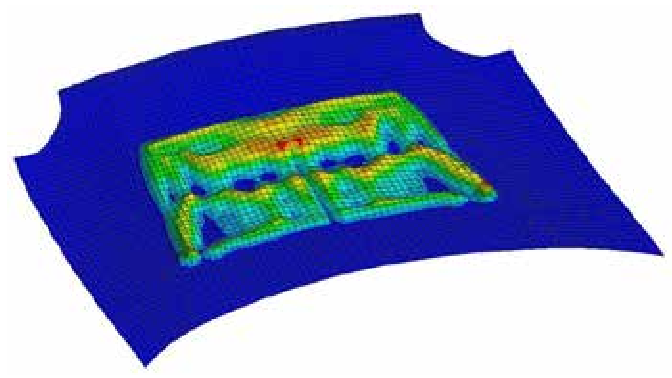


Fig. 6 Topography Optimization application

Freeform Optimization

Freeform design is also a special type of shape optimization. The intention is to increase the design freedom with little effort from the user. GSAM will automatically create perturbations for grids on selected surfaces. During optimization process the grid locations are changed to achieve a better design. Optionally the user can apply fabrication requirements such as symmetries, extrusion or grid fraction. Freeform optimization can also be used by automatically linking adjacent grids to reduce computational cost.

Topometry Optimization

Topometry is an element by element sizing optimization capability. This capability is to increase the design freedom for problems where the user has flexibility for design changes. Typically topometry design can be used to find the thickness distribution of plates. It can also be used to find the best elements to keep from a pool of elements. Fabrication constraints such as symmetries and extrusion can be imposed on topometry designed regions. Coarse topometry option is available to reduce computational cost.

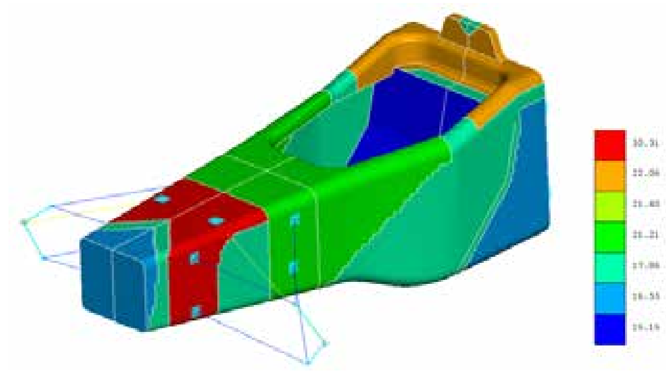


Fig. 7 - Topometry Optimization application

Sizing Optimization

Sizing optimization is to design the dimension of the elements. In current GSAM implementation, the thickness of shells can be designed. GSAM will automatically create design variables associated with the thickness of the shell. Sizing optimization is typically used at detailed design stage to gain more material savings and improve design performance.

Mixed Optimization

Topology optimization can be combined with sizing, topometry, topography, and/or freeform design. Optimization problems may have both topology designable regions and parametric designed entities at the same time. All response types may be used in any optimization setup (i.e., topology-only, parametric-only or mixed).

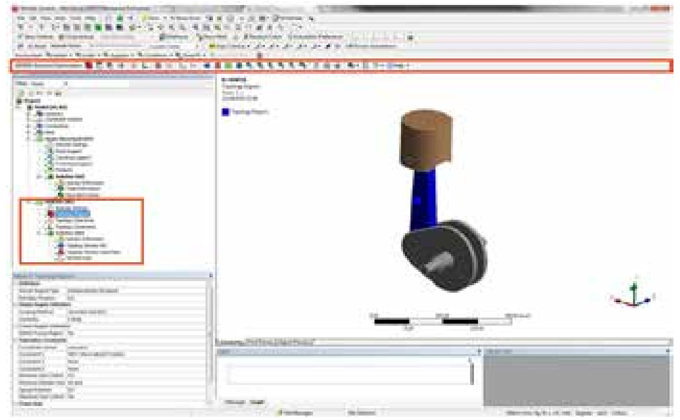


Fig. 8 - GSAM in Workbench

The Interface in workbench allows easy and fast creation of structural optimization data, efficacious post-processing, clean and clear solutions and to export optimized geometry in STL and IGES, by pressing the buttons on the GSAM toolbar.

Proceeding from left to right on the GSAM toolbar (see Fig. 7), is in fact possible to define the design regions and fabrication constraints, the objectives, the constraint response functions, to check the iterations during the run, to post process the results and export optimized geometry and mesh.

In conclusion GSAM is a very effective tool that achieve light and performing structures at the same time within ANSYS Workbench, respecting the manufacturing limits, quickly and with little effort.

GSAM (Genesis Structural Optimization for ANSYS Mechanical) is supported in Europe by EnginSoft. For more information:
Martina Guidi, EnginSoft
m.guidi@enginsoft.it

Fig. 7 GSAM toolbar

Recurdyn establishes new Paradigms in Flexible Multibody Simulation

Flexible bodies in MultiBody Simulation

In the past years, the use of MultiBody Simulation (MBS) in industry has progressively grown. This approach is used to investigate both kinematics and dynamics of moving mechanisms, which are composed by multiple bodies interacting with each other through joints and contacts. MultiBody Simulation is the recommended numerical method to quickly complete the following tasks:

- design of mechanisms for motion control (e.g. cams, links, guides)
- check of functionality and performance assessment (e.g. interference check, speed and acceleration analysis)
- estimation of joint loads and internal reactions in transient conditions, in order to choose actuators, brakes and other power devices.

The most basic MBS approach idealizes the system components as rigid bodies. Although sometimes this assumption is fairly acceptable, a high number of applications cannot be virtualized, ignoring the body compliance. Flexibility affects the value of the participating inertia, influences the application points of loads and changes the way the kinetic energy is dissipated in the system. For all of these reasons, different numerical methods have been developed in order to introduce flexible bodies in MultiBody Simulation. Although general guidelines cannot be formulated, flexible MultiBody Simulation is recommended in, at least, three situations:

- when external loads have frequencies close to structural ones (resonances);
- when the system undergoes high speed dynamics and vibrations affect the requested outputs;
- when the calculation of stress and strains in transient conditions is a mandatory output of the study.

For the types of outputs it provides, a flexible MultiBody Simulation can be assimilated to a transient Finite Element Analysis. The big difference is in the numerical formulation of the two problems, which makes MultiBody a little less precise, but much (much) faster. Moreover, the MultiBody approach, even when it includes flexible bodies, it keeps its distinctive natural connectivity with control system design, pneumatics, hydraulics, and electronics.
RecurDyn, the premium multibody software from FunctionBay Korea, is a key technology in this scenario. It implements two alternative technologies for flexible body modeling. This paper compares the two approaches and highlights the advantages of each one of them.

Case study

The multibody model of an excavator is chosen as reference case study. The simplicity of this example helps in maximizing the differences between the modeling approaches.

A series of revolute joints connects the arm bodies in a single kinematic chain going from the bucket to the cabin. The latter body is then connected to the vehicle base through one more revolute joint with vertical axis. Three groups of hydraulic cylinders control the arm configuration. Each actuator consists of a piston and a cylinder, coupled together by a translational joint. Both ends of each actuator are linked to the excavator structure by means of revolute/spherical joints. The overall kinematic scheme is fairly simple and represents the degrees of freedom of the actual excavator.

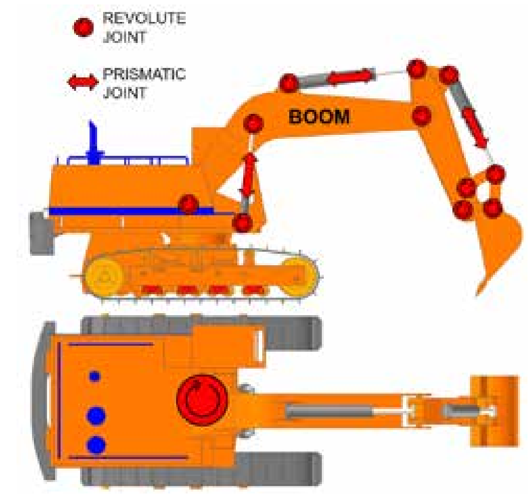


Figure 1 – Kinematic scheme of the excavator model

The model is initially built using rigid bodies only. This step is useful to check the appropriateness of joints, drivers, motion functions and contacts. The flexibility is applied to selected bodies in a second phase. In general, it is not necessary to switch all bodies to flexible and, more important, it is not necessary to use the same approach for all flexible bodies. In our example, we will convert just the excavator boom to a flexible, which is the main part of the machine arm.
The simulation reproduces a standard working cycle of the excavator. The bucket approaches a target object (not modeled), digs it, transports it, unloads it and finally moves back to the initial position. All tasks are completed in about eleven seconds. The motion is obtained by governing the lengths of the hydraulic actuators and the angular position of the revolute joint between base and cabin.

Rigid Body Dynamics

The dynamic analysis with a rigid multibody excavator model returns the outputs of Figure 2. The plots show the reaction forces and torques of the revolute joints connecting the boom to the rest of the model.

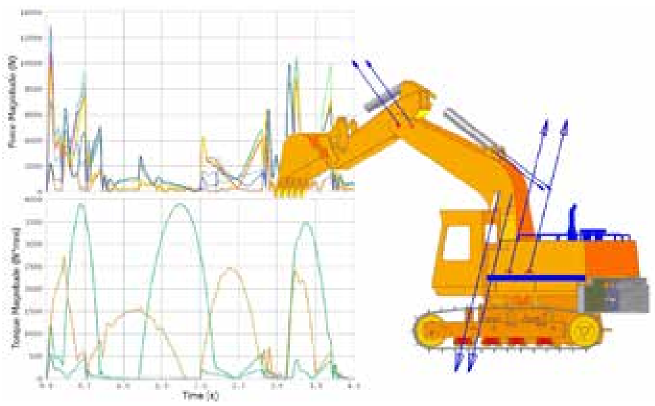


Figure 2 – Analysis results of the rigid body dynamics analysis. Reaction forces and torques of the joints that link the boom to the excavator system

The rigid body simulation confirms that all motion laws are correctly defined. Moreover, the internal reaction loads are perfect inputs for a structural assessment, which has to be carried out using an external FE code. Since the load histories have multiple peaks, generally not synchronized, a complete structural analysis of the boom requires to repeat the FE calculation multiple times. The Figure 3 shows the meshed boom (shell elements) loaded with one of the force and moment set extracted from the multibody simulation.

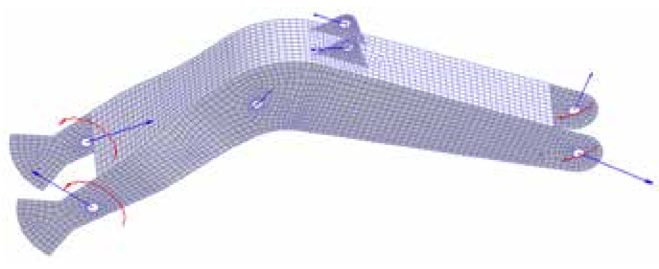


Figure 3 – FE analysis of the excavator boom. The FE model is loaded with forces and torques calculated through rigid MBS

The loads coming from a rigid multibody analysis are averagely higher than the true ones, because the moving inertia is overestimated. Accordingly, the power demand of the actuators is overestimated. From a structural point of view the procedure is conservative, although it is not easy to rate how much. For mechanisms moving at slow speed (with respect to the first natural frequency of the structure), the two step procedure is fairly applicable. For high speed dynamics, the use of rigid bodies easily leads to excessive loads and, therefore, excessive sizing of the components. Another weak point of this procedure is that two separate codes (Multi-Body and Finite Element) are required. There is always a risk of error in the load data transfer, especially because the multibody loads have spatial components over moving bodies.

Flexible Multi-Body Simulation is an interesting alternative to this traditional approach, which improves the quality of the results and, at the same time, makes the whole calculation process straightforward and easier.

Reduced Flex Technology (RFlex)

In RecurDyn, the Reduced Flex method coincides with the well-known and widely accepted Craig-Bampton approach. The method was developed at the end of '70 in the aerospace industry, to reduce the overall size of large FE models. The theory assumes that the response of a flexible body in static (and dynamic) conditions can be represented by a linear superposition of several mode shapes, which is why this approach is also known as Component Mode Synthesis. By doing so, the initial meshed body is translated into a mathematical object whose unknowns are the linear multipliers of the base modes. This results in an evident reduction in the number of unknowns. This theory has been expanded and adjusted along the years, but the original rules are still valid and applied to create flexible bodies in almost every modern multibody software.
Figure 4 graphically describes the RFLex approach in RecurDyn. The modal basis, which will numerically describe the body flexibility, is created by combining two sets of structural modes: the fixed interface vibration modes and the so-called constraint modes. The result is a mathematical object whose unknowns are the multipliers of the orthonormalized modes.
All of these operations require an external Finite Element code, which provides the tools for meshing the geometry and for performing the necessary FE analyses. As all of the competitor software, RecurDyn can import RFLex data from ANSYS and NASTRAN. However, it also includes both an internal mesher and an internal FE solver that make it possible to prepare the RFlex data without the need of an external FE code.

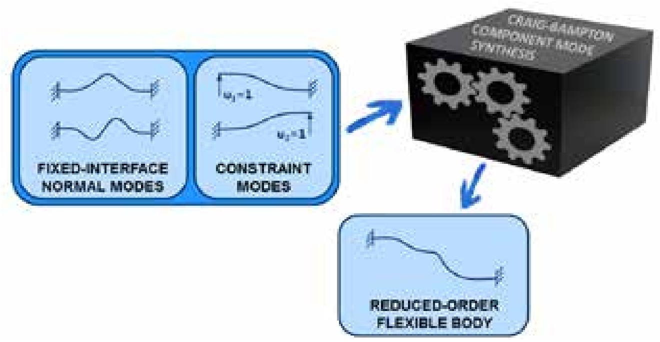


Figure 4 – A graphical overview of the Craig-Bampton approach for flexible bodies modal reduction

In our model, the Reduced Flex method is applied to the excavator boom. First the geometry is meshed to obtain a FE model composed by shell elements. The thickness is set in accordance with the starting CAD, while the material is set to steel (linear isotropic properties). In order to establish a physical connection between the joints and the geometry of the flexible body, RecurDyn creates Force Distributing Rigid (FDR) elements (Figure 5). Each FDR element has a master node at the joint center, and a spider of links (could be thought as rigid beams) connected to the scoped geometry. The master nodes always have 6 degrees of freedom (3 rotations and 3 translations), so that the deformations induced by both forces and moments over the structure can be correctly accounted.

First the fixed-interface modal analysis is performed on the FE model. For our problem, we limited the calculation to the first 30 eigenvalues and eigenvectors. There is not a general rule to fix this number;

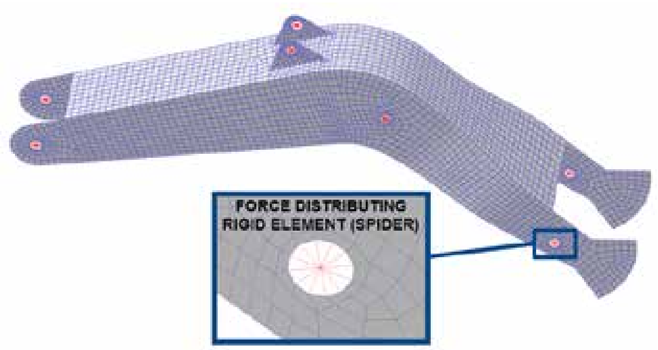


Figure 5 – FE model of the boom (shell elements). The interfaces to the joints of the excavator model are set up with FDR elements (spiders)

normally the user evaluates the mode frequencies and chooses the number in accordance to the higher phenomena he would like to see in the simulation. Then, 48 constraint modes are calculated. This number is obtained by multiplying the number of master nodes (the boom has 8 joints) by the number of Degrees Of Freedom (DOF) of each master node (6). Each constraint mode consists in a static analysis where a unit displacement is applied to a single DOF, while the remaining ones are kept to zero.

After the pre-calculation phase, we get a total of $30 + 48 = 78$ modes. These modes are combined together and orthonormalized to generate the RFlex modal base. Some of the modes included in this mathematical object are shown in Figure 6.

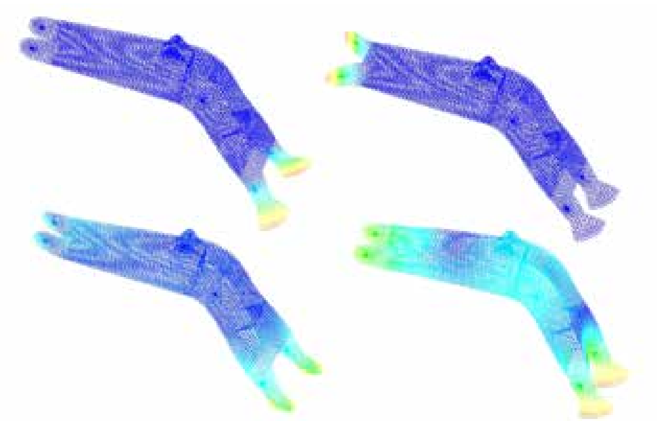


Figure 6 – Four of the 78 boom deformation modes calculated through the use Craig-Bampton component mode synthesis method

Once the RFlex boom is ready, it is incorporated in the excavator model (joints are automatically connected to the master nodes) and the MultiBody Simulation is performed as usual. Besides the results we got earlier from the rigid multibody simulation, the model now outputs also stresses and strains. These structural quantities are available over the entire boom extension, over all simulation time. This makes possible to easily identify where and when the most critical stress state occurs.

Figure 7 shows the distribution of the equivalent Von Mises stress over the boom deformed body, at the instant where it reaches the maximum value. In our example, the peak is much lower than the material strength. The highest stress is even lower than the fatigue limit, excluding any type of structural problem for this structure.

By watching the animation of the results, it is easy to observe oscillations of the excavator arm that were not visible in the rigid

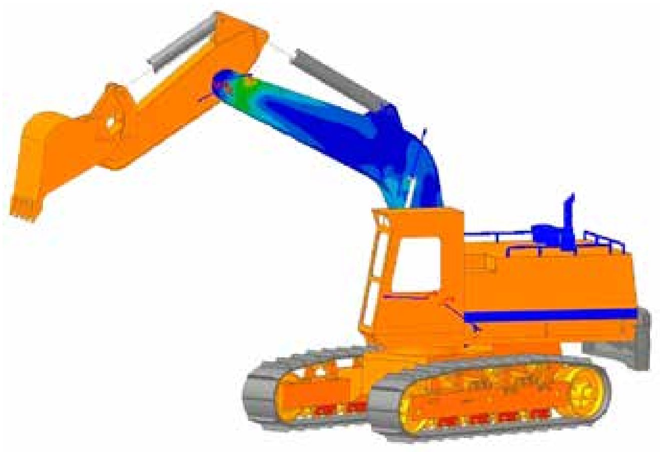


Figure 7 – Analysis results of the ReducedFlex analysis. Reaction forces and torques of the boom joints are displayed and Von Mises equivalent stress is plotted on the boom deformed body.

multibody results. This is a realistic behavior, which can be easily observed in a true working excavator.

FullFlex Technology (FFlex)

Despite being very well performing, RFlex technology has intrinsic limits that make it unusable in several applications. First, the linear behavior is acceptable for small deformations only. Second, it is almost impossible to properly describe the effects of contacts through a master node interface.

In order to overcome these limits, FunctionBay has introduced in RecurDyn an advanced approach for flexible body modeling. The FullFlex (FFlex) technology is a simplified implementation of the Finite Element formulation. While RFlex method speeds up the solution by reducing the number of unknowns, the FFlex method is based on a smart simplification of the equations that describe the coordinates of all mesh nodes.

A FFlex model keeps all of its native DOFs, but the solution time is terrifically reduced. Every MultiBody Simulation performed with RecurDyn FullFlex technology is equivalent to a transient Finite Element analysis. For this reason, this advanced approach has been called Multi-Flexible-Body Dynamics (MFBD).

The FullFlex technology breaks all limits of the ReducedFlex one. It calculates the structural response of structures undergoing large rotations and large displacements. It also manages large deformations, with the ability to simulate a non-linear behavior of the material. It also makes possible the definition of contacts over the bodies (solids, shells, beams) with no restrictions.

From a numerical point of view, FullFlex models have a noticeable number of DOFs. However, thanks to both the smart formulation of the equations and the power of the hybrid solver, RecurDyn assures very fair computing time.

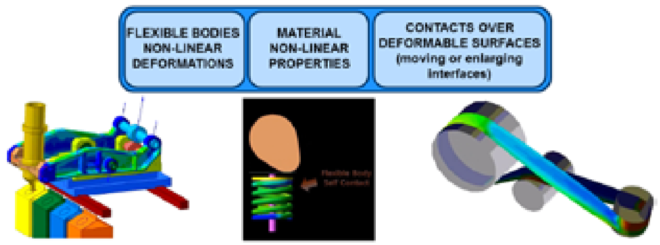


Figure 8 – Benefits that the FullFlex technology offers in multibody simulation.

In our excavator example the RFlex technology worked just fine and returned reliable structural results. However, the areas where the joints are located are unavoidably affected by modeling approximations. Indeed, the Force Distributing Rigid elements make them rigid and recreate an unrealistic stress state.

In order to highlight the benefits of the FullFlex technology, we have modified the excavator model and, in particular, the flexible formulation of the boom body. A new solid mesh is created, with specific refinements applied on the connecting areas (Figure 9). FDR interfaces are kept only for the connections of the actuators, whereas non-linear contacts are set between holes and pins on the two boom ends.

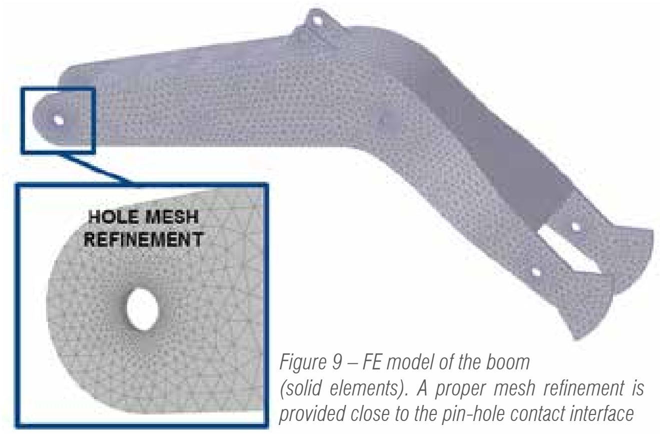


Figure 9 – FE model of the boom (solid elements). A proper mesh refinement is provided close to the pin-hole contact interface

The dynamic analysis of the excavator with this new model returns almost the same structural results we got previously. As shown in Figure 10 the maximum value of the Von Mises stress is still lower than the fatigue limit. From a different perspective, this confirms that Reduced Flex and Full Flex technologies are both reliable.

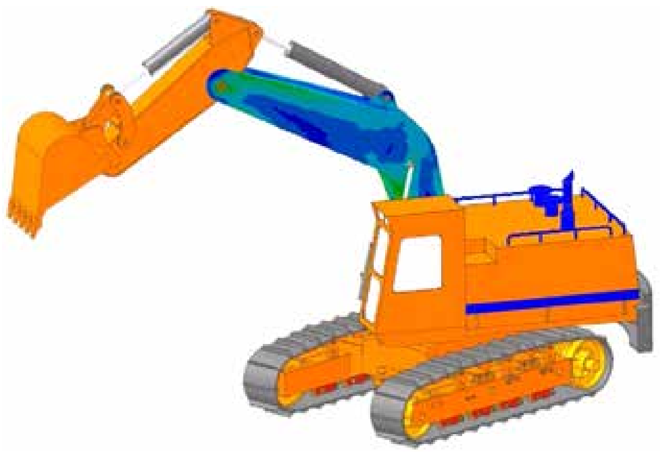


Figure 10 – Analysis results of the FullFlex analysis. Von Mises equivalent stress is plotted on the boom deformed body

The Figure 11 provides a significant detail of the Von Mises equivalent stress distribution at the interface between boom and dipper stick. There is a clear subdivision of the hole in two regions: one region is stressed by the pin pressure while the second region is almost unloaded. Such result cannot be captured using the Reduced Flex approach.

Comparison of methods and conclusions

The table in Figure 12 points out the main differences between the three possible approaches to model a mechanical system in RecurDyn. The complexity of the model (and the calculation time) grows going from left to right, as the number of unknowns grows as well. The user should always choose the approach that returns the desired output with the minimum computational effort.

The most significant advantages of the RecurDyn Full Flex approach can be summarized as follows:

- modeling of body connections in a very realistic way, without introducing any local stiffening spider;
- large deformations, large rotations and large displacements are natively taken into account;
- it is possible to use both linear and non-linear materials;
- there is no limit in the use of non-linear contacts between flexible bodies. They can be set between solid, shell and beam elements;
- the simulated dynamics of flexible bodies is exact, because it is no longer the output of a transfer function based on selected vibration modes (Craig-Bampton approach).

For all of the above reasons, RecurDyn is the most powerful and most versatile software in the market, designed to perform flexible multibody simulation.

For more information:
Fabiano Maggio, EnginSoft
f.maggio@enginsoft.it

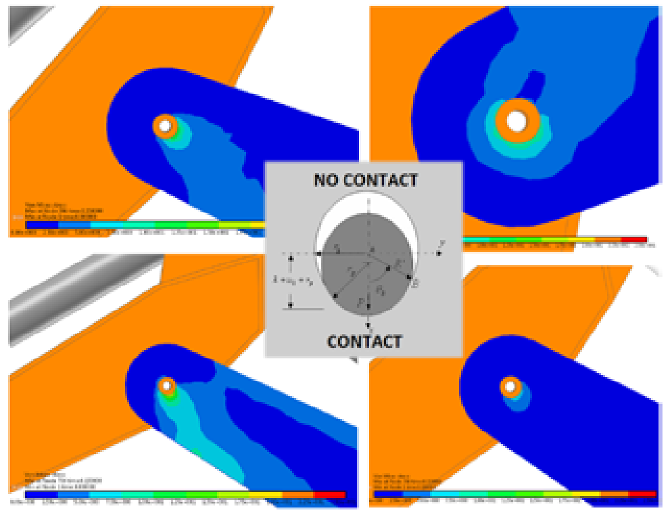


Figure 11 – Detail of the pin-hole contact interaction between boom and dipper stick (Von Mises equivalent stress plotted on boom deformed body)

	Rigid Body	ReducedFlex Body	FullFlex Body RECURDYN
INERTIA TENSOR, MASS	FROM 3D GEOMETRY	FEA PRE-CALCULATION	FROME ELEMENTS AND PROPERTIES
SPATIAL DIMENSIONS	INFINITE	FE MESH	FE MESH
CONNECTIVITIES (JOINTS, FORCES)	ANYWHERE	LIMITED TO STATIC EXTERNAL NODES (WITH SPIDERS)	AT ALL NODES
ELASTICITY	-	REDUCED TO LINEAR COMBINATION OF EIGENMODES	FULL STIFFNESS MATRIX AVAILABLE
DEGREES OF FREEDOM	6	# OF ACTIVE EIGENMODES + 6 RIGID BODY DOFs	6 x # OF NODES (SHELL, BEAM) 3 x # OF NODES (SOLID)
DEFORMATION	-	LINEAR	NON-LINEAR
DYNAMICS	EXACT	DEPENDING ON # OF MODES	EXACT
CONTACTS	RIGID-RIGID	RIGID-RIGID (LOCAL STIFFENING)	RIGID-FLEXIBLE FLEXIBLE-FLEXIBLE
MATERIALS	-	LINEAR	LINEAR, NON-LINEAR
STRESS ANALYSIS	-	LOCALLY CRITICAL	EXACT

Figure 12 – Comparison between the capabilities of the three applied modeling approaches.

modeFRONTIER 2016

The innovative Optimization Environment with Modular, Profile-based Access

The philosophy behind modeFRONTIER 2016 benefits your entire organization by reducing complexity, improving efficiency and cutting development time. modeFRONTIER 2016 provides teams with enhanced flexibility in performing advanced optimization, process integration and automation. User Profiles respond to different expert needs and facilitate effective workload balance within design teams. Each profile focuses on specific sets of design optimization and integration practices to boost engineering productivity.

New User Profiles enable multidisciplinary engineering practices to consolidate specialized expertise and streamline teamwork by allocating software resources where needed. Depending on the step of the engineering problem at hand, it is now possible to access different functionalities within the same installation through dedicated modules (modeSPACE and modePROCESS) or directly in modeFRONTIER, according to the profile of the user.

- modeSPACE: the design space environment is now also available as a standalone application, modeSPACE, that enables efficient license and role management within teams. This module includes the sophisticated set of modeFRONTIER tools for data analysis and investigation of problem characteristics both in the post-processing and in the pre-optimization phase.
- modePROCESS: modePROCESS is an independent desktop application intended for describing processes in the form of graphical workflows, that specifies which parameters and simulations are required to solve an engineering design problem.

modeFRONTIER 2016, what's new?

Improved direct integrations node capabilities

ANSYS WB Parametric Pack
The ANSYS Workbench integration node now supports the ANSYS Parametric Pack licensing scheme. modeFRONTIER users will be able to launch multiple concurrent design evaluations with a single set of keys, i.e. without checking out additional Workbench licenses, taking advantage of their own Parametric Pack.

ABAQUS CAE
Previously, modeFRONTIER could run ABAQUS CAE simulations only through an indirect integration, whereas the ABAQUS direct integration node could only extract data from ABAQUS output databases. The ABAQUS node functionalities have now been extended also to support the run of CAE model simulations.

Improved algorithm

No-settings pilOPT v2
Improvements to respond to an even wider variety of engineering design problems, more balanced and efficient exploitation of computational resources, effective handling of problems with discrete variables, increased performance with single-objective problems.

Simplified algorithms' parameters list for basic user
Hide the complexity of algorithm settings (DOE, Scheduler, RSM), all available parameters are visible in the "Expert mode" instead of showing up by default - less confusing for less experienced users.

Improved post-processing functionalities

Automatic RSM Training Mode
All actions required for the RSM training can be performed in a single panel, going straight from data to metamodels with less clicks and default parameter settings.

Sensitivity Analysis Tool
SS-Anova is now a Standalone Tool available in the Design Space and each screening can be saved as a function. Screening functions can be used in the RSM wizard to improve the training.

Further improvements and new functionalities will be available in the new modeFRONTIER 2016
Don't miss the opportunity!

For more information:
Francesco Franchini, EnginSoft
f.franchini@enginsoft.it

PICK YOUR PROFILE

Monitoring the results of your simulation team and need to be always updated on the status? DATA VIEWER

Facing time or computational resource barriers or have a scarce knowledge of the design problem characteristics? OPTIMEAZY

Dealing with very complex, multidisciplinary problems? MULTI DISCIPLINE DESIGN

Are you an expert in Design Space Exploration and RSM-based what-if analysis? DATA EXPLORER

Are you the team domain expert focusing on single-discipline simulations? SINGLE DISCIPLINE DESIGN

In charge of integrating and automating multiple disciplines in a single workflow? AUTOMATION DESIGNER

mF 2016 modeFRONTIER

	profile	functionalities	interface mS	mP	mF
DATA SPACE	DATA VIEWER	Analytics charts [VIEW MODE]	✓		
	DATA INSPECTOR	Analytics charts [VIEW AND EDIT]	✓		
	DATA INTELLIGENCE	Analytics tools	✓		
	DATA EXPLORER	Analytics tools Design Space Exploration	✓	✓	✓
PROCESS & OPTIMIZATION	AUTOMATION DESIGNER	Workflow [CREATE, EDIT, RUN] Integration package medium		✓	✓
	SINGLE DISCIPLINE DESIGN	Workflow [CREATE, EDIT, RUN] Analytics tools Design Space Exploration Integration package medium	✓	✓	✓
	MULTI DISCIPLINE DESIGN	Workflow [CREATE, EDIT, RUN] Analytics tools Design Space Exploration Robust Design & MCDM Integration package large	✓	✓	✓
	OPTIMEAZY	pilOPT workflow [CREATE, EDIT, RUN] Integration package small	✓	✓	✓

LEGEND

Analytics charts:
statistics charts

Analytics tools:
statistics charts
MVA (clustering + SOM*)
RSM

* Self-Organizing Maps

Design Space Exploration:
DOE workflows
Sensitivity Analysis

Integration Packages:*
SMALL: 1 any CAD/CAE node
MEDIUM: 3 any CAD/CAE nodes
LARGE: 10 any CAD/CAE nodes

* Choose your nodes every time you set a project

Robust Design & Decision Making:
MORDO
MCDM

pilOPT workflow:
DOE
pilOPT
1 Integration Node

EXTRA Toolbox
› CAP
› GRID
› EASYDRIVER

NPE Packages
› SMALL (64)
› MEDIUM (512)
› LARGE (1024)

What's New in Flowmaster V7.9.5?

Flowmaster V7.9.5 delivers a new 3D Visualizer which is accessed through the new Airside Visualizer and Segmenter (AVS) module. The tool provides a more modern look and feel as well as improving the user experience. Enhancements have also been applied to the Live Charting in order to reduce its impact on simulation time.

Live Charting

The new enhancements implemented for the Live Charting have reduced its impact on simulation time. In addition, in order to minimize the simulation time, Live Charts can optionally be conditionally updated after an interval and not every time-step.

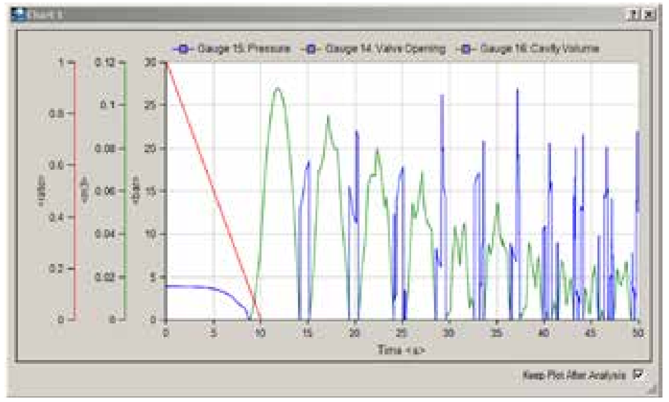


Figure 1 - Live Charting enhancements reduce its impact on simulation time

The Visualizer

The Visualizer is a 3D graphical tool that enables to view the spatial arrangement of the components comprising the cooling pack. The cooling pack may include components such as the grill, stacked heat exchangers (charge air cooler, oil coolers, condenser, radiator, etc.) fans and losses, with air inlet and outlet boundaries.
For example, the spatial arrangement of the simple cooling pack represented in Figure 2 is visualized in Figure 3. In the Visualizer, spheres represent boundary conditions, such as Flow or Pressure Sources. Components such as Condensers, Heat Exchangers, etc. are shown as cuboid objects. Users can set the geometrical data for each component, i.e. its position in 3D space, its height, width and thickness, etc. on the relevant component data form. Colour and transparency level can also be set for each component.

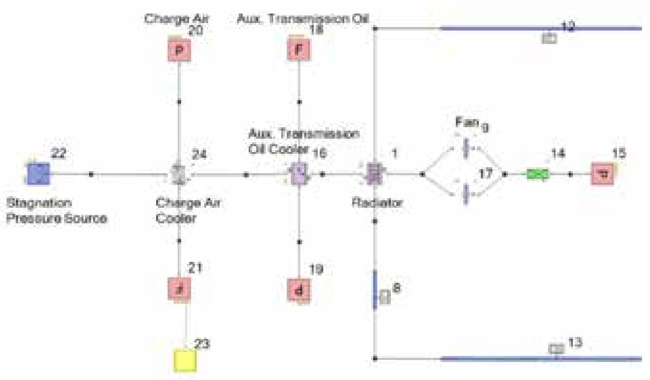


Figure 2 - Schematic representation of a simple cooling pack in a Flowmaster network

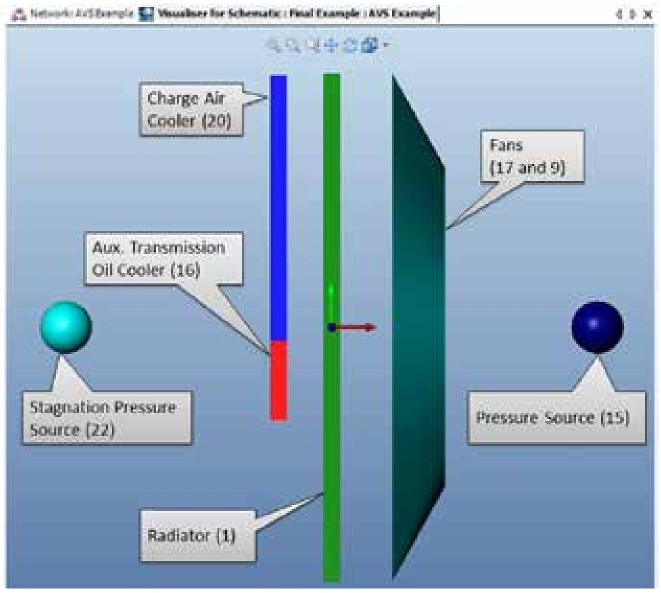


Figure 3 - Spatial arrangement of the cooling pack represented in Figure 2 as seen in the new 3D Visualizer

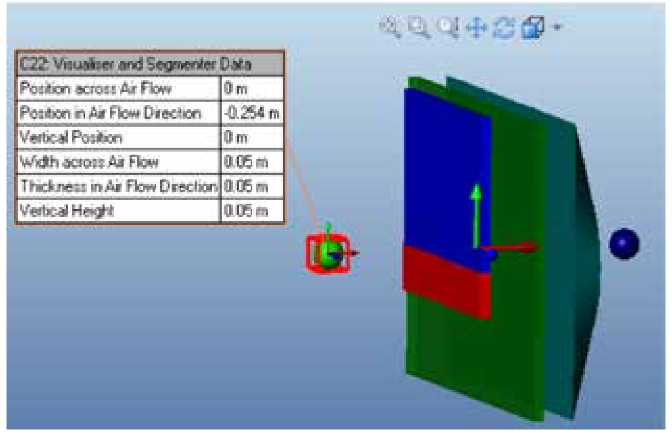


Figure 4 - New callout displaying selected component information are implemented within the new 3D Visualizer

In the new 3D Visualizer the user experience is improved by means of a number of enhancements:

- the new toolbar implemented within the 3D Visualizer make the interaction simpler and faster;
- with the new dynamic update, the 3D scene reacts and update simultaneously when a change is made in the Visualizer and Segmenter input data form;
- when a component is selected in the 3D Visualizer, new callouts with component information are displayed (Figure 4);
- with the new dynamic explode it is possible to dynamically increase the space between the components in order to inspect the 3D arrangements in a better and easier way (Figure 5).

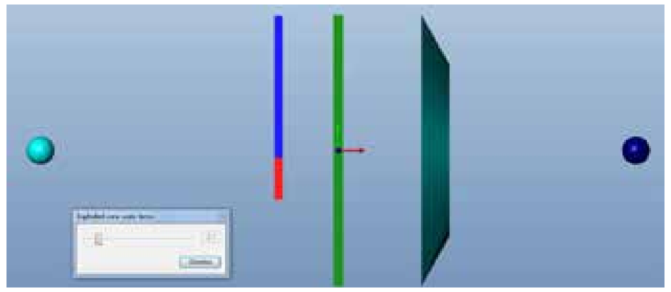


Figure 5 - New dynamic explode for better and easier inspection of the 3D arrangement of the components

The Segmenter

Real life cooling packs may have complex 3D arrangements and be located behind grills and fenders. This implies that the heat exchangers may be partially or totally hidden by other components so that different zones of the heat exchangers receive air at different velocity and temperature. It is crucial to account for all of this in order to accurately compute the cooling pack performance.
The Segmenter identifies groups of Heat Exchangers, and any other segmentable components, which are adjoining and in series in the air flow path. It uses the geometric data to identify a segregated air flow path for each overlapping and non-overlapping segment of all the segmentable components in each contiguous group. It then creates the appropriate number of segments within each of these components and applies the upstream boundary conditions based on any given inlet distribution (Velocity and/or Temperature) in a Stagnation Pressure Source.
With regard to the orientation of the hot stream flow in the original components, it creates either horizontal or vertical hot stream flow paths of consecutive segments in series, and connects these parallel flow paths to each other at the edges of each segmented component. For example, if the segmentation produces six segments in a Radiator, in two horizontal rows of three for a horizontal flow radiator, the Segmenter will connect these as two rows of three in series and will connect the ends of these rows in parallel as shown in Figure 6.

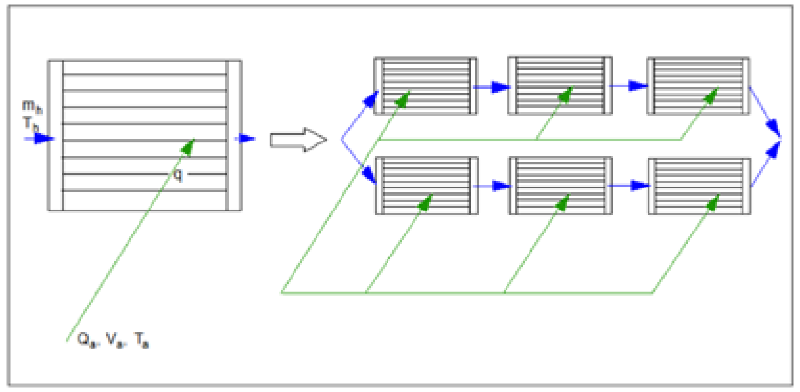


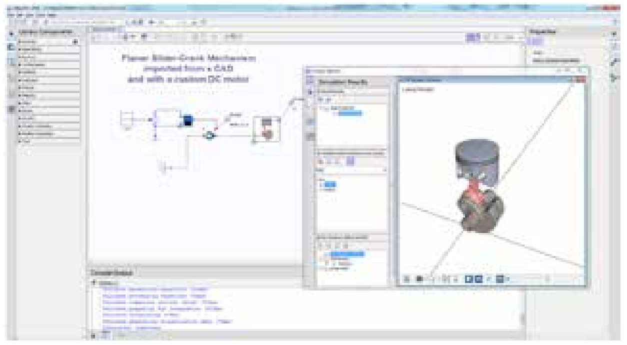
Figure 6 - Example of a segmented radiator

For more information:
Alberto Deponti, EnginSoft - a.deponti@enginsoft.it

Discover Maple and MapleSim with EnginSoft webinars

Since May 2016, EnginSoft began to introduce Maple and MapleSim through webinars, during which the key features and capabilities were highlighted in terms of their industrial applications.These webinars are available for viewing at the URL:
http://www.enginsoft.it/form/downloadwebinar.html
The first webinar introduced Maple and MapleSim capabilities with a special focus on industrial applications, exemplified by two simple but significant sample cases.
In the first example we show how to leverage Maple in order to create a small worksheet on thermodynamics, presenting the same concepts and techniques employed in more complex industrial scenarios (such as HVAC systems and thermodynamic cycle efficiency calculation). In this example we show how a typical enthalpy-pressure chart can be plotted (for example, R134a and MDM fluids); these charts are useful for calculating the behavior or the performance of ORC (Organic Rankine Cycle) systems.
In the second example we analyze the behavior of a dynamic mechanical system with MapleSim and the CAD Toolbox: these two tools allow the user to import a CAD model directly into the MapleSim environment and analyze the mechanical performances of the system in an efficient manner.
The figure shows a MapleSim model representing a system composed of a mechanical system and an electrical motor, in which the piston is commanded by the electrical motor. The physical model of the piston is imported directly in MapleSim through the CAD Toolbox.

For further information about Maplesoft products:
Manolo Venturin, EnginSoft - m.venturin@enginsoft.it



EnginSoft KEY partner in EU Exascale project

This Horizon2020 FET-HPC ExaNeSt project develops and prototypes solutions for some of the crucial problems on the way towards production of Exascale-level Supercomputers

EnginSoft has been involved as one of the key engineering partners for the development of the new European Exascale platform through the "ExaNest" project. This three year project which started in December 2015, will develop, evaluate, and prototype the physical platform and architectural solution for a unified Communication and Storage Interconnect and the physical rack and environmental structures required to deliver European Exascale Systems.

The consortium brings technology, skills, and knowledge across the entire value chain from computing IP to packaging and system deployment; and from operating systems, storage, and communication to HPC with big data management, algorithms, applications, and frameworks. Building on a decade of advanced R&D, ExaNeSt will deliver the solution that can support exascale deployment in the follow-up industrial commercialization phases. Using direction from the ETP4HPC roadmap and soon to be available high density and efficiency compute. The ExaNest consortium will model, simulate, and validate through prototype, a system with:

Fig. 1 - Ultra dense immerse cooling technology provided by Iceotope

- High throughput, low latency connectivity, suitable for exascale-level compute, their storage, and I/O, with congestion mitigation, QoS guarantees, and resilience.
- Support for distributed storage located with the compute elements providing low latency that non-volatile memories require, while reducing energy, complexity, and costs.
- Support for task-to-data sw locality models to ensure minimum data communication energy overheads and property maintenance in databases.
- Hyper-density system integration scheme that will develop a modular, commercial, European-sourced advanced cooling system for exascale in ~200 racks while maintaining reliability and cost of ownership.
- The platform management scheme for big-data I/O to this resilient, unified distributed storage compute architecture.
- Demonstrate the applicability of the platform for the complete spectrum of Big Data applications, e.g. from HPC simulations to Business Intelligence support. All aspects will be steered and validated with the first-hand experience of HPC applications and experts, through kernel turning and subsequent data management and application analysis.

The effort "said Gino Perna head of HPC and Software integration in Enginsoft", is unique in seeking to advance ARM64+FPGA architecture as a foundational "general-purpose" exascale platform. We are deeply involved in porting CFD and Engineering application to this platform and optimizing performances in order to achieve the goals". The chosen platform and innovative liquid cooling system will be key points in providing extremely dense systems and energy to performance very attractive for next generation HPC centers.

With the core technologies of its partners and collaborators, ExaNeSt is anticipated to complete its first straw man prototype in 2016, a full prototype in 2018, and will inevitably leave a trail of innovation in its path.

Visit: http://www.exanest.eu

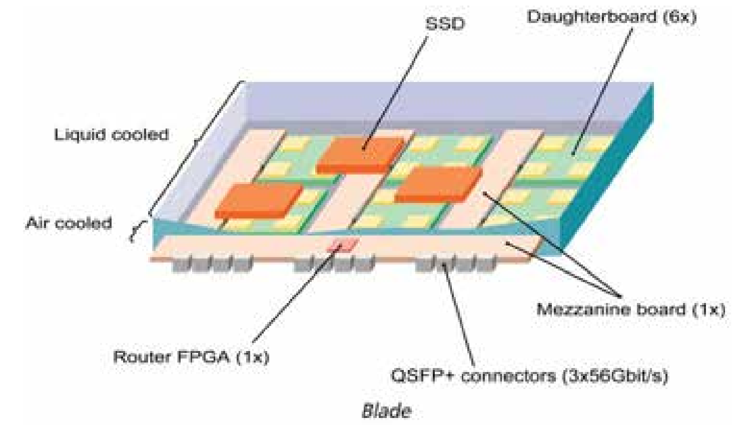


Fig. 2 - Exanest Blade design

MUSIC Project at High Tech Die Casting 2016

The MUSIC project (MUlti-layers control&cognitive System to drive metal and plastic production line for Injected Components) will be one of the sponsors of the High Tech Die Casting 2016, that will take place in Venice on 22nd-23rd June.
Since 2002, HTDC Conference has become a key-event for the international industrial and academic community involved in casting processes of Aluminium, Magnesium and other non-ferrous alloys. The HTDC Conference has been steadily growing in quality, impact and participation in the following conferences, organised in 2004, 2006, 2008 and 2012. Today, Die Casting production is facing new challenges: from eco-sustainability and efficient energy usage to optimization of product properties, from alloys' properties improvement to design of lightweight components. Only an "open minded" approach, resulting in a high capability of being innovation-driven, integration-oriented and implementation-ready, will make Die Casting foundries successful in a international arena dominated by competition.
In such perspective, this high-profile event will include session totally dedicated to MUSIC, a sort of mini-course delivered by the project 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 new project book "Smart Control and Cognitive system applied in HPDC foundry 4.0, 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

INTERNATIONAL
CAE CONFERENCE

SAVE THE DATE

17th - 18th
OCTOBER
2016

PARMA
PAGANINI CONGRESSI
ITALY

FOSTERING A SYNERGISTIC ENVIRONMENT
IN ENGINEERING SIMULATION

Striving for innovation excellence?

CALL FOR PAPERS IS NOW OPEN!

*A great opportunity to submit an abstract
and share your expertise!*

WWW.CAECONFERENCE.COM