Newsletter

Simulation Based Engineering & Sciences

Year 14 n°2 Summer 2017

Improving the development of farming equipment using CAE technologies

Investigate different configurations and increase the overall **switcher performances**

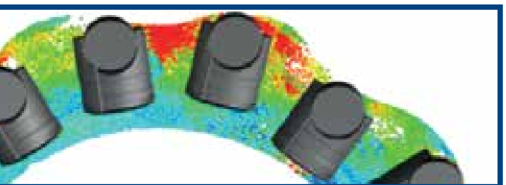
Reliability-based RDO Using Polynomial Chaos Expansion for **Aeronautics Applications**

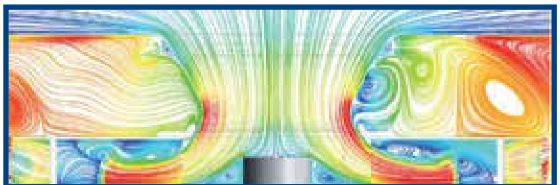
Visualization of **Oil Lubrication** in the Transfer Case and the Transmission

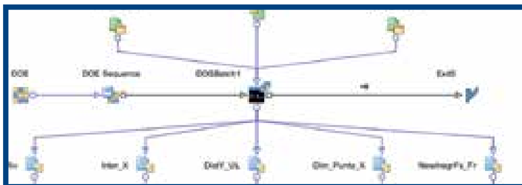
Increase the efficiency of of **fans and compressors** at Boldrocchi

Reliable **hydraulic direct drives** for improved performance

Optimum Design of Diesel **Ship Engine Silencer**







FLASH

The 4th Industrial revolution is upon us, otherwise known as Industry 4.0. From the mechanization of the 1st industrial revolution in the 18th century, we have experienced the benefits of mass production (2nd revolution) and computer automation (3rd revolution). However, we are now moving into the 4th, Cyber-Physical systems, where physical systems have a virtual copy and encompasses areas such as the internet of things and cloud computing.

Many companies have already taken the necessary steps to support their Industry 4.0 strategy by adopting advanced data management or innovative simulation software into their design process.

The advantages of Industry 4.0 to transform the foundry process is evident on page 48. It reveals how a digital transformation can provide real time data to make adjustments, allowing for improvements in production efficiently. A complete adoption of an Industry 4.0 strategy could be a real game-changer, allowing costs to be minimized and reducing time-to-market to gain competitive advantage.

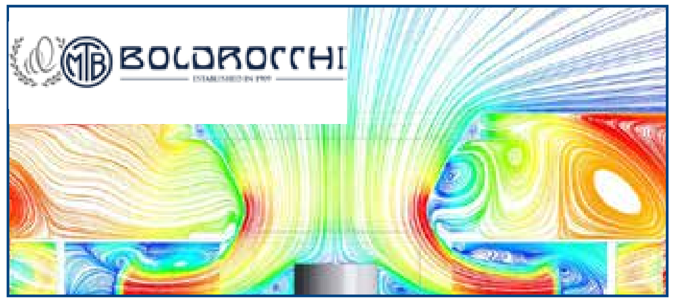
Where oil plays an important role and conditions of its behaviour is often inspected on a physical product, significant changes to the product design at this stage is near impossible due to the costs involved. Univance details the rewarding use of simulation on page 22 for the understanding of such oil behaviour in a gearbox by introducing Particleworks and RecurDyn.

As demonstrated in the articles simulation can act as a catalyst towards an Industry 4.0 strategy. At the upcoming International CAE Conference I am excited to further explore this 'soul' of industry 4.0.

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 14 n°2 - Summer 2017

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

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: l.cunico@enginsoft.com

RESPONSIBLE DIRECTOR
Stefano Odorizzi
ART DIRECTOR
Luisa Cunico - l.cunico@enginsoft.com
PRINTING Grafiche Dal Piaz - Trento

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

Interview with Diego Barone, R&D Project Leader in Vin Service

Vin Service was founded in 1976 from oenologist Riccardo Guadalupi and Daniela Gennaro Guadalupi, based on the revolutionary idea of supplying quality wine, in large quantities and fast… on tap.

First Italian manufacturer of draft wine dispensers, it steadily grew and became one of the world's leader in the dispense technology, serving large international brands such as Heineken, Pilsner Urquell, Coca-Cola, PepsiCo, Carlsberg, Budweiser and many others. Today, the core business of Vin Service is represented by dispensing equipment industry, from soft drinks to draught beer, and by cooling systems for beverages.

Vin Service is constantly innovating and improving its processes to always provide its clients with the best made-in-Italy customized dispensing solutions. Its expert R&D division is equipped with mechanical engineers, CAD 3D designer, 3D printers and a certified in-house lab for testing. Plug&Play dispensers for disposable PET kegs and innovative Peltier Technology for draught beer and soft drink towers are some of Vin Service main bespoke solutions for the beverage industry.

1) When did you first introduce the use of simulation in your company?

In 2011 we had to develop a new product for a very important customer and, to be more competitive, we decided to replace low-pressure die cast brass with injection molded plastic. Budget for moulds was so big to easily justify doing simulation-based design and virtual prototyping upfront. After this first experience, it has been easy to adopt such techniques every time a project was big enough or complex enough for us.

2) What was the main reason for introducing simulation?

Trial-and-error and over-design are old-school approaches that have no place in modern super-competitive globalized industrial markets. Our company is growing fast and had to transform to be accepted by global customers as a real world-class manufacturing company.

3) What kind of products are you using simulation for?

Currently we're mainly doing validation of mechanical design and exploration of design alternatives according to customized load-cases or norms and standards. For example, we have recently explored a design alternative for cost reduction, reducing the number of parts while maintaining the same structural resistance of a beer countertop cooler.

4) How does simulation affect your design process?

As of today, we are still at work to transform our design process in a true simulation-based approach. We currently partner with external CAE providers, and in particular with EnginSoft, on a case-by-case basis. Replacing the old empirical approach with computer simulations allowed us to significantly speed-up and simplify our design process, enabling a 80% reduction in development time (from 12 weeks to 2-3 weeks) and a 70% reduction in project costs.

5) Do you have any plans to extend the use of simulation in your company?

We're offering simulation-based design in every proposal we issue to our customers, in order to increase demand and justify the introduction of internal resources. Next step will be expanding the use of structural and CFD analyses on a regular basis in our R&D workflow to accelerate the feasibility assessment of new designs.

6) What is the contribution that EnginSoft can give you?

In EnginSoft we can find experts with a cumulative experience of many years in all industries and we rely on them according to the variable needs of our customers. During the past 6 years EnginSoft has been a strategic partner for the innovation of our products, from the improvement of mechanical performances of our beverage dispensers to the introduction of new cost-effective materials.

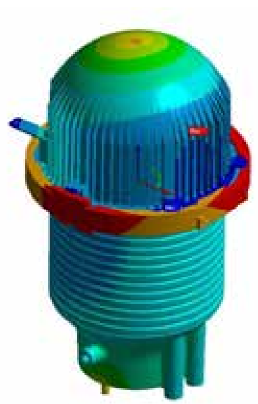
7) What value did simulation bring to your company?

Faster time-to-market, better in-depth analysis of design alternatives, virtual prototyping, process and product cost reduction, these are only some of the numerous advantages that simulation brought to our company. Switching to a simulation-guided approach for example allowed us to introduce the new Plug&Play maintenance-free dispensers for PET kegs.

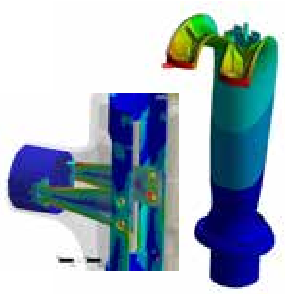
This innovative technique enhanced our company perception by our customers and provided us a competitive advantage which helped Vin Service winning highly rewarding projects.

Vin Service and EnginSoft: a six-year history of innovation

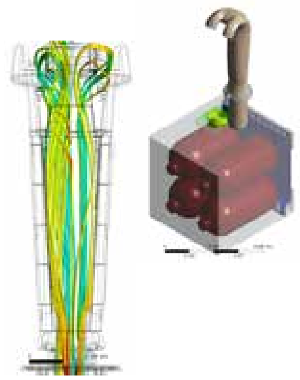
1 The first collaboration started in 2011, when a new CO_2-free dispenser was added to Vin Service product portfolio. This new countertop dispenser adopts a pressure chamber to dispense cold beer without external CO_2 supply. The technological challenges, mostly related to pressure inside the chamber, were promptly solved thanks to simulations, which guaranteed the correct structural requirements for given pressures.



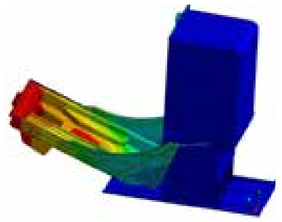
2 From 2011 to 2016, 5 different column-shaped beer dispensers were analyzed and improved by EnginSoft, focusing on the mechanical resistance of the products. Innovative solutions were obtained by replacing the metallic body with fiber-reinforced polymers, ensuring both higher mechanical characteristics and lower material costs.



3 In addition, CFD simulations of the whole dispenser (from barrels to fins) were performed. The analyses quantified the effects of air heating on the thermal quality of the beer. Thermal efficiency of the dispenser was virtually tested and improved by reinforcing the insulation in selected areas, by reshaping the column, or by guiding the correct choice and positioning of the fans.

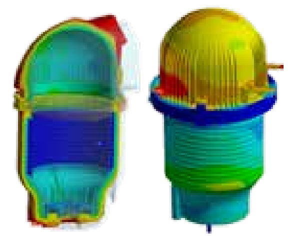


4 In 2014, structural simulations were applied to a new plug&play countertop cooler with recyclable PET kegs, so that the product could meet customer's requirements. This activity evaluated the stability of the refrigerator and determined the correct sizing of geometries and the choice of materials to minimize stresses and deformations of the structure.



5 In 2016 the plug&play countertop cooler with recyclable PET kegs was further improved by integrating mechanical, esthetical and thermal performances into a single piece, i.e. the door. This innovation allowed to significantly reduce production costs by minimizing the number of parts to produce and by lowering the complexity of the assembly.

6 The most recent innovation is the improvement of the original CO_2-free countertop beer dispenser. Structural simulations were expanded to include thermal and fatigue analyses, plus the effects of aging of plastic materials. High stress regions were identified and modifications in the insulating layer were suggested to improve thermal efficiency.



Improving the development of farming equipment using CAE technologies

The deep integration of ANSYS and Spaceclaim in the design process of Maschio Gaspardo SpA allows the company to evaluate in advance the performances of its products, improving them to ensure quality

The company
Maschio Gaspardo SpA is a leading company in the agricultural equipment market, deploying one of the widest varieties of farming implements worldwide. The production focuses on equipment for soil preparation, seeding, haymaking and crop protection.

Spraying equipment and its engineering challenges
"Crop protection" is an expression that defines all the activities aimed to defend cultivations from threats menacing (directly or indirectly) the quality and profitability of the harvest. Usually, these activities sum up in the distribution of phytosanitary products over the fields. The distribution is performed with specific implements called sprayers, mainly built following a common design: a central tank holds the mixture to be dispersed, while lateral folding beams (called spraying booms) sustain the tubing that carries the fluid over the crop. The extension of the booms allows the coverage of a wide area of work (well over 30 meters).
Although necessary to save the crop, the use of phytosanitary products is to be limited to a minimum: in fact, it is an expense for the farmer and causes the reduction of quality of the harvest, being it exposed to chemicals.
To ensure that with minimum quantities of phytosanitary the full protection of the crop is still reached, structural and dynamic performances of the spraying equipment are crucial: the stability of the booms ensures homogeneity and accuracy of the dispersion, reducing the need for over-spraying. Lightweight booms with low inertia allow for the use of active control systems, that continuously and efficiently modify the geometry to better follow the profile of the soil, holding a constant spraying height. Furthermore, the reduction of weight and inertia is extremely important to grant maneuverability both on the road and on the field, enabling higher working speed. The direct consequences are the reduction of working times and the improvement of response time against threats, especially on large cultivations.

Therefore, the structural and oscillatory behaviors of the spraying booms define the capability of standing out amongst the competitors in the sprayer market. The study and optimization of these factors become a priority in the design process. To ensure that the new, top of the line spraying boom developed by the company will meet the performance requirements defined and expected by the customers, ANSYS software has been deeply integrated in the design process, helping designers from the first design steps to the testing and validation process of the final product.

> "Using ANSYS and this new design procedure, Maschio Gaspardo SpA avoids the use of the old "trial & error" approach that required the production of a large number of prototypes. Now we are able to save design and manufacturing time, manpower and money, producing and testing only a limited number of already well optimized prototypes" says Natalino Dorigutto, Project Manager at Maschio Gaspardo SpA

The new spraying boom
On the new spraying boom project, specific constraints posed by the market have defined the general layout of the structure, consisting of a series of sections joined with hinges and hydraulically actuated.

Figure 1 - On the left, MY15 spraying boom in its final design. On the right, M16 boom in an advanced stage of development

The benchmark for the study has been defined considering a similar model of boom from the previous year. The new boom's performance needs to at least match this baseline, while providing a significant reduction in weight. For sake of simplicity, the old model will be referred as "MY15" and the new one "MY16", accordingly to the years of development (Fig. 1).
FEM models of both MY15 and MY16 have been built using similar simplifications and procedures: the aim was to obtain a significant comparative result, but at the end of the project the results also proved to be compatible with the absolute measurements.
Geometries have been defeatured following standard practices, taking advantage of almost all Spaceclaim capabilities. Midsurfaces and beams were extracted, negligible features were removed and geometries were rearranged to match the configurations required for the study (Fig. 2).

Figure 2 - On the left, original geometry for MY16 as received from CAD. On the right, geometry after the defeaturing process

Connections, joints, local refinements and point masses have been introduced in the ANSYS Mechanical model transferring named selections and coordinate systems from Spaceclaim to ANSYS: this is a significant advantage in terms of productivity, since a rearrangement of the geometry in Spaceclaim no longer requires the user to manually adjust all the features in ANSYS Mechanical once the geometry is updated. In a similar manner, meshing is built using named selection to automate the process of region selection in case of geometry modification.
To model weld joints of adjacent components, the "share topology" feature has been widely used, ensuring a quick and computationally light representation. In this way, also, information on the welds are not lost: in fact, local shell-to-solid submodelling is always available to further investigate the welded region if required, enabling to focus all the computational resources on a small portion of frame.
Hydraulic actuators have been represented either with rigid bodies or with springs and dampers elements (set up to match experimental data). The representation is chosen accordingly to the requirements of the tests to be performed on the physical prototype at the end of the project (Fig. 3).

Figure 3 - Example of mesh obtained on the defeatured geometry prior to the simulation of the whole assembly

Stage 1. Torsional analysis
An important but sometimes overlooked problem is the torsion of the boom during the opening motion. This is a problem that lies outside of the working conditions mentioned before, but it is of the utmost importance to provide the farmer with the feeling of a robust implement. A "weak" boom, that twists more than expected, will not be accepted (Fig. 4).

Figure 4 - Effect of low torsional stiffness on older boom models

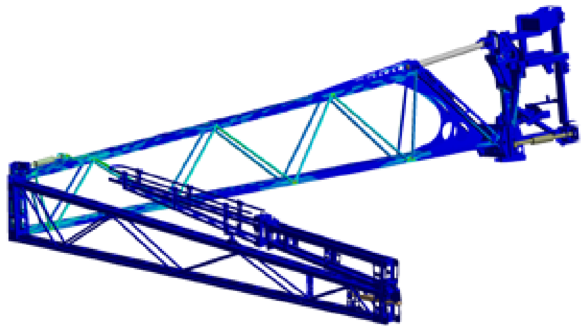


Figure 5 - Example of torsional analysis performed"

The Mechanical model shows that the new MY16 boom, reinforced with local plates, manages to maintain the performances of the older model, thus satisfying the requirements of the project (Fig. 5).
The defeaturing performed moving from tubular components to beam elements allows the designers to receive information on axial and bending loads acting on the tube, defining safety factors against elastic instability. Tube diameters have been reduced where possible and corrective actions have been enforced where needed.
This analysis also informs on the loads on the various hinges, allowing the definition of contact pressure and the correct sizing of bushings.

Stage 2. First bending analysis

For safety purposes, booms are required to have the so-called "safety joint" at a certain distance from the tip: if the boom hits an obstacle (namely a tree, a pole etc.) between the safety joint and the tip, the applied force/displacement will disengage the joint. Regulations require the boom to sustain the hit undamaged below a certain speed. To compare the safety coefficients of the worst-case scenario hit, static analyses are set up for both MY15 and MY16: static structural analyses are considered suitable being low the speed defined by regulations (Fig. 6).

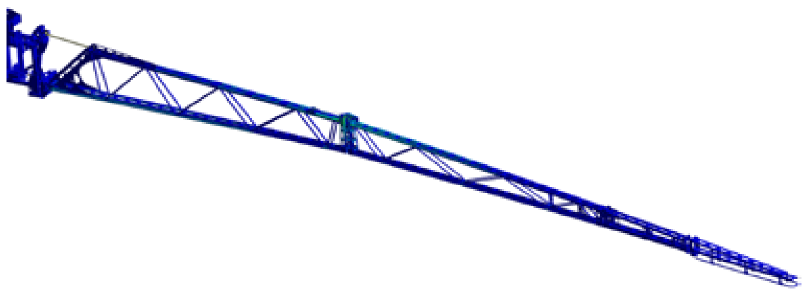


Figure 6 - Example of bending analysis to evaluate the stress state in case of activation of the safety joint

The analyses once again assess the quality of the MY16 design, which shows safety coefficients in the range or above those of MY15.
As a side result, the analysis provides with a first indication of the bending stiffness against actions in the direction of forward motion.
Moving from Stage1 to Stage2 of the study, features set up in Spaceclaim and ANSYS to automatically adjust and rebuild the models really come in handy, requiring only a small number of inputs from the user to adapt the model to the new configuration.

Stage 3. Second bending analysis

Spraying booms hold spraying nozzles, which are all required to have the same ground clearance along the boom. On a perfectly flat field, this produces a constant spraying pattern beneath the implement. If the boom is subjected to large vertical displacements along its path, however, the spraying pattern becomes irregular. Also, active control systems are hindered, being the actual position of the nozzles different from that assumed by their software. It can easily be thought of adjusting the height of the nozzles to counteract the effects of the deformed shape, but this is not sufficient: the position will be correct in steady conditions, but during field use a high value of stiffness is required to avoid noticeable deformations when the sprayer hits uneven tracks.
Therefore, stiffness in the vertical direction must be evaluated. This is carried out in this stage (Fig. 7).

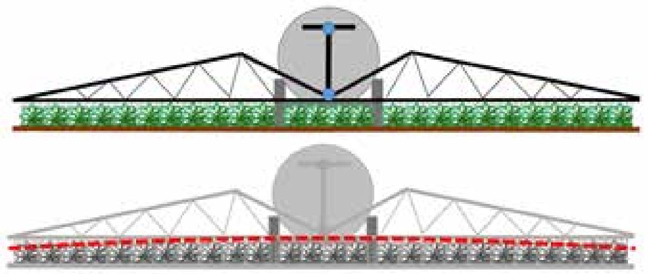


Figure 7- Bending effect on the vertical plane. On top, ideal condition. Below, actual configuration

The Mechanical model predicts a vertical stiffness for MY16 close to that of MY15, which satisfies the imposed requirements: spraying nozzles are held on an almost horizontal line over the 36 meters of width of the sprayer.

Stage 4. Modal analysis of the structure

Analyses performed to this point aimed to define the behavior in static situations: stiffness and mass interacted but an overall view of the relationship between them is lacking. To address this, modal analyses are performed on MY15 and MY16 booms. Modal analyses sum up the results of interest in a convenient way, providing with "comparable" indexes (natural frequencies and mode shapes) while also informing the team of designers about possible resonance problems.
Usually, a problem with the comparison of modal results

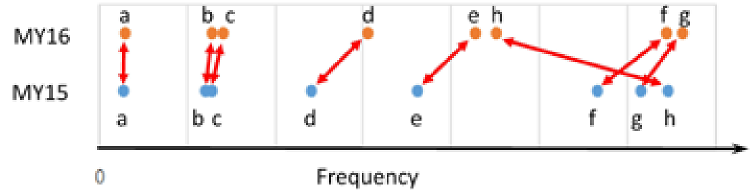


Figure 8 - Mode shapes and associated frequencies: comparison between MY15 and MY16

on complex structures consists in recognizing the same mode shape on different geometries. This process becomes extremely tricky when modes clearly distinct on one geometry tends to become mixed on another.
In this case, MY15 and MY16 are different products that share many core features of the base structure: this leads to similar mode shapes and to ease of comparison of the output (Fig. 8).

The results are promising: moving from MY15 to MY16, every mode shape is shifted to higher frequencies. Considering the frequencies of common excitations acting on the system, it can be stated that this shift has a positive effect on the boom's behavior: the appearance of detrimental mode shapes will be reduced, leading to a more stable boom. A significant exception is that related to the torsional mode: its appearance (mode "h") is anticipated to lower frequencies, but this is

Figure 9 - Overshoot of an older model of spraying boom during the braking action

not considered as a problem being the torsional mode hardly subjected to any excitation during field use.

Stage 5. Transient simulation of a braking action

To this point, the static and modal results have been extremely useful to compare MY15 and MY16, but the behavior in a real-case dynamic scenario is still unknown. To evaluate this situation, a braking simulation is performed. This condition is chosen being it the most demanding, from a structural point of view, during field operation. It's also extremely useful to evaluate the quality of the spraying action: the backward and forward motion associated to the booms during braking is one of

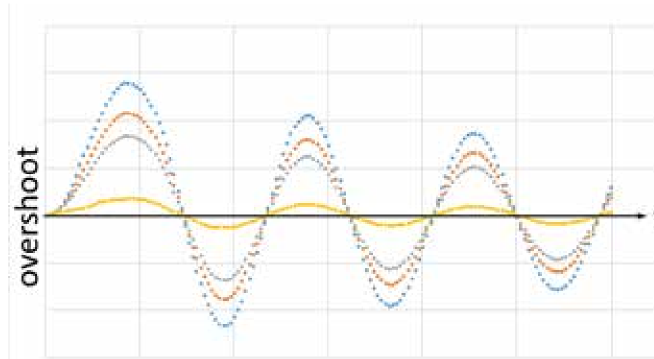


Figure 10 - Overshoot of a boom in the transient analysis. Different colors correspond to different positions on the boom, picked to evaluate how different nozzles are affected by the braking action

the major causes of over-spraying on the crop, situation that needs to be avoided reducing the amplitude of the movement and its duration (Fig. 9)

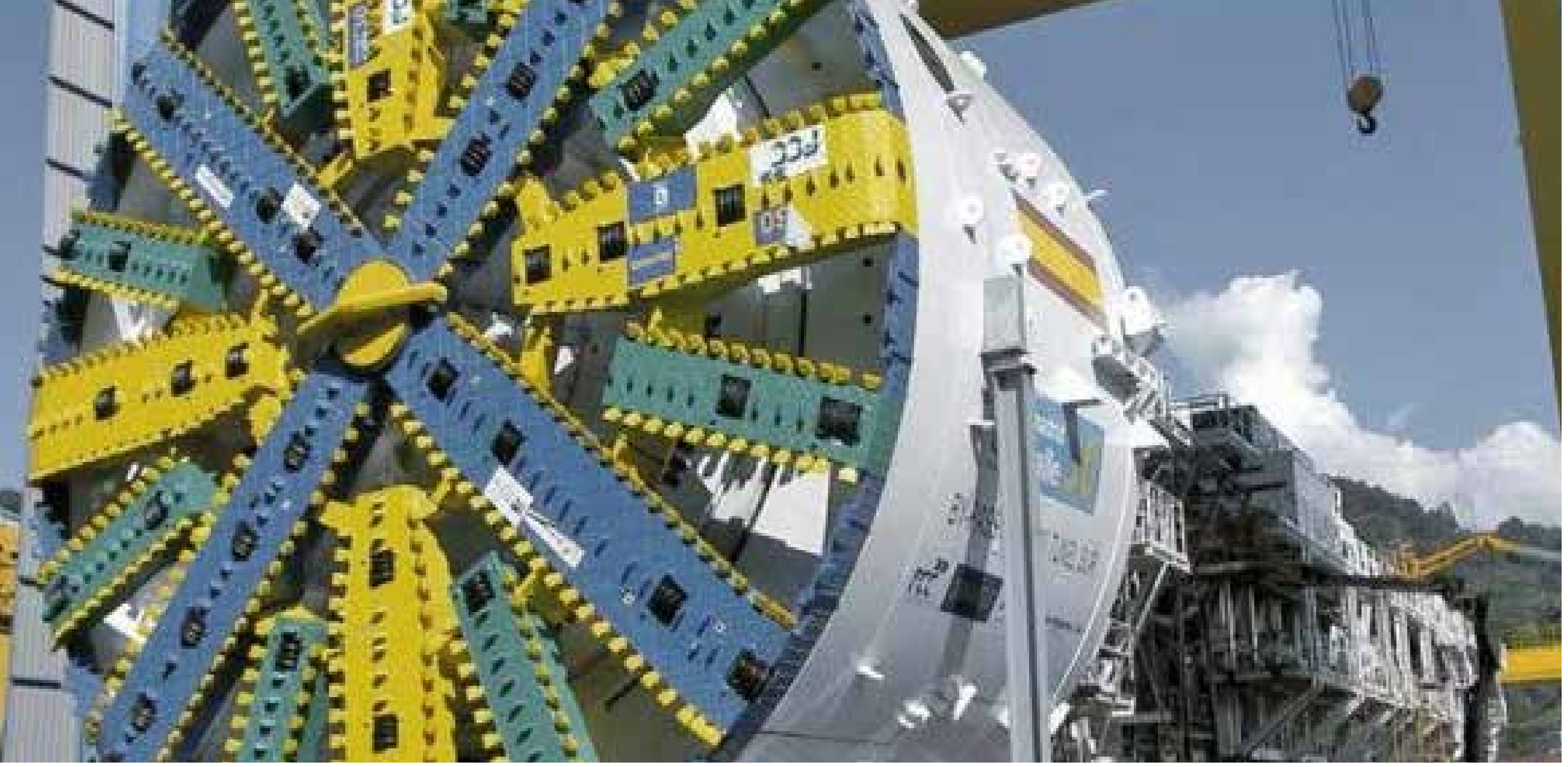
The simulation is run assigning to the structure an initial velocity, then a deceleration is imposed to the joints connecting the booms to the main frame of the sprayer, accordingly to experimental data (Fig. 10).
Once again, MY16 proves to be an efficient design. The deceleration input excites only the first mode shape in the horizontal plane (which is similar to the deformed configuration observed in Stage 2), and MY16, which has a much lower inertia than MY15, reduces the overshoot of about 20% compared that of the old model (the overshoot is defined picking as a reference the main frame of the sprayer).

Conclusions

The new design of the boom has been thoroughly evaluated during the project, using static, modal and transient analyses. The FEM team and the design team of Maschio Gaspardo SpA interacted profitably to address weaknesses emerged and to further improve the weight reduction and the safety coefficients of the product.

The final design, transferred to physical prototype, has behaved accordingly to the analyses (thus validating the numerical work) and has satisfied the project requirements: the new boom is lighter than the old model and exhibits significant improvement in stability. The wide use of FEM analyses reduced the need for prototypes and shrank the development time compared to old projects. During the creation and elaboration of this product, performed along the renewed R&D guidelines of Maschio Gaspardo SpA, Ansys acquired such an importance that it became an irreplaceable tool of the design process. Its conjoined use with Spaceclaim has further improved its capabilities, allowing for fast transfer of geometry, defeaturing and analysis, even with complex and large assemblies.

Violato Valentino
R&D Dept. Maschio Gaspardo SpA



Reliable hydraulic direct drives for improved performance

Torque and losses prediction, calculation of oil residence time and understanding the flow path are key factors during the development phases of a hydraulic motor

This article present the study of a radial piston hydraulic motor using a mesh-less CFD methodology called Moving Particle Simulation and available in the Particleworks software. Torque and losses prediction, calculation of oil residence time and understanding the flow path are key factors during the development phases of the motor. The geometrical complexity and the moving parts of the system are easily handled by the mesh-less CFD methodology.

Rexroth Bosch Group The Drive & Control Company

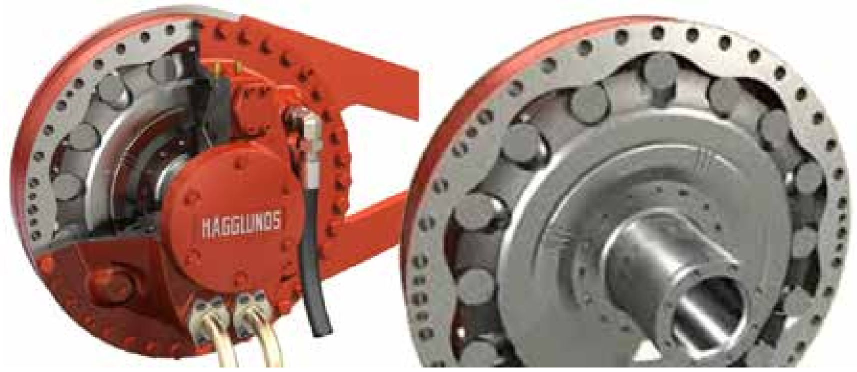


Figure 1 - Cam and piston arrangement of the Radial Piston Engine

Introduction

Bosch Rexroth Mellansel AB delivers complete hydraulic drive systems for the industry, mainly where low rotational speed and high torque are required. Bosch Rexroth drive systems are applied in several industrial sectors for heavy duty applications, from the mining and materials handling to the cement industry, from the Oil&Gas to the pulp and paper industries and many others.

One part of the drive system is the Hägglunds Compact motor. This is a radial piston hydraulic motor with an outside cam arrangement. The housing of the motor is completely filled with hydraulic oil that aids in cooling and lubrication. Extra cooling is added in high power applications by flushing oil through the housing with an external pump.

Reliability and low environmental impact are important customer needs and are characteristic for Bosch Rexroth products. Continuous work on reducing losses is important to fulfill these needs. One source of losses in the motor is the resistance to the rotating motion of the shaft called churning losses. These losses are almost negligible in low speed applications, but at high speed, they will contribute to substantial heating of the motor casing. Churning losses come from drag by pushing oil in front of each roller in a similar way as in a rolling element bearing.

Understanding the oil flow in the piston engine, predicting the cooling efficiency and the churning losses is of paramount importance to improve the performance of the drive system and this has been pursued by Bosch Rexroth in the last years by building mathematical models that try to represent the oil flow inside the engine.

The geometrical complexity of the system and the motion of the shaft and cylinders make the modelling of the complete engine practically unfeasible for traditional fluid-dynamic techniques, based on computational mesh. That is why, so far, only partial models of portions of the system has been created and used to study the local oil behaviour.

Recently Bosch Rexroth decided to try a new methodology to simulate the oil flow inside the entire radial piston engine. This methodology is the Moving Particle Simulation (MPS) available in the Particleworks software. The MPS is a methodology to solve the Navier-Stokes equations, where the fluid volume is discretized by fluid particles instead of grid elements.

The mesh-free nature of the MPS allows easy modelling of complex geometries like the entire radial piston engine, with an easy definition of the motions of the shaft and cylinders and the simulation of the interaction between the moving bodies and the oil flow.

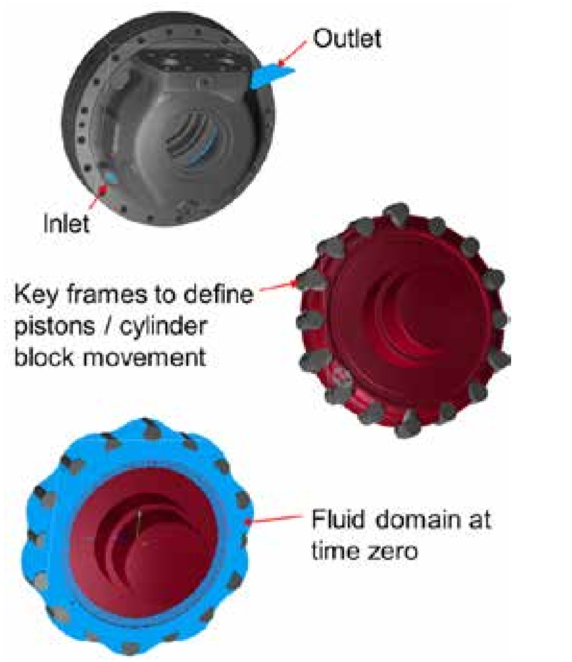


Figure 2- CAD objects of the Radial Piston Engine in Particleworks

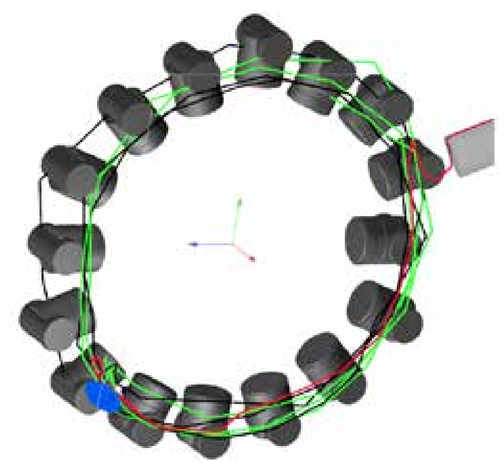


Figure 3- Individual oil particle tracking in Particleworks

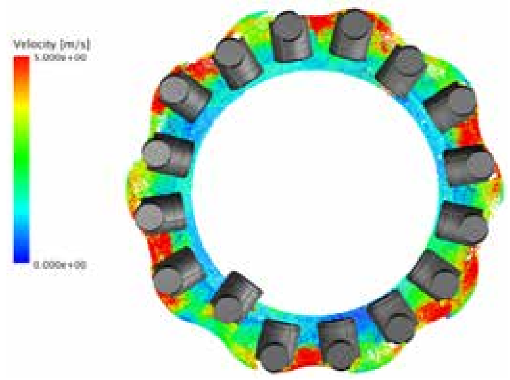


Figure 4- Velocity field between the piston and cam arrangement

In an initial test phase of the MPS methodology Bosch Rexroth built the model of the complete radial piston engine, from the oil inlet to the outlet and performed a number of simulations with promising results.

The 3D model was able to give a deep insight into the system behavior by taking into account every single detail of the engine (no assumptions, reductions or simplifications), and by giving quantitative outputs like churning losses, oil residence time, pressure and velocity in every single point of the engine.

Despite the complexity of the radial piston engine, the time and engineering effort to build the complete MPS model is quite small. The process starts with the direct import of the housing, cylinder

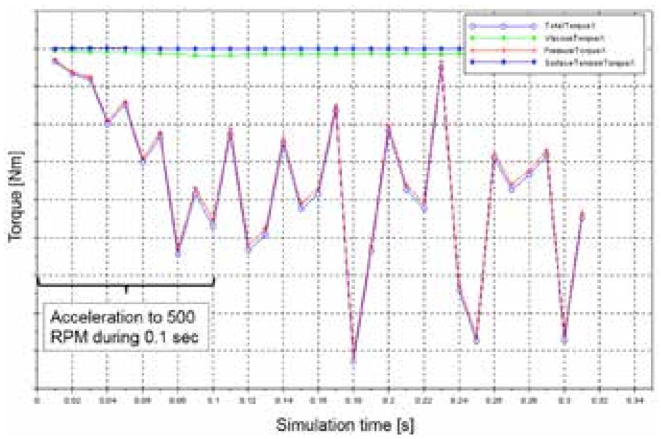


Figure 5 - Torque on the 16 pistons + Cylinder block

Figure 6 - Applications of the Hägglunds compact motor

block and pistons from the CAD into the Particleworks software. Neither geometrical simplifications nor mesh generation are needed.
Key frame scripts define the rotation of the cylinder block and the reciprocating movement of the pistons, while oil properties are defined in the software together with the physical and numerical models, that allow for the solution of the oil motion (Navier-Stokes equations).
The global time needed to create the MPS model starting from the CAD files is less than 1 hour. Running one transient simulation for a total time of 0.5[s] takes between 0.5 and 3.5 days depending on the level of accuracy and on the available hardware.
The level of accuracy in the MPS method is simply defined by the particle size that is applied to discretize the oil volume. Reliable results for the radial piston engine can be obtained with particle size of 1 mm, while very accurate results can be found decreasing the particle size down to 0.5 mm, that allows to solve in a better way the oil fluxes in all the small gaps of the system.
Globally the time to build and run the MPS model of the radial piston engine is between 1 and 4 days. This means that the use of Particleworks and the moving particle simulation really fits in the normal design process and can support the daily design activities of the engine.
Moreover the comparison of different designs of the engine or the investigation of how operating conditions like rpm, flow rate, oil properties affect the performance of the engine can be done virtually, by the use of fluid-dynamic simulation.
The benefit and value of this methodology is not only related to time and money savings in the development of the engine, but it is mainly associated to the knowledge that can be acquired by monitoring what happens inside the system.
In the figure 3 the oil particle location vs. time is plotted in the Radial Piston Engine. By tracking several oil particles the residence time and the circulation time of oil can be calculated, which gives indications of the temperature evolution.
In the figure 4 the velocity field between the pistons and cam arrangement is plotted.
In the figure 5 the total torque (churning losses) from surface tension, viscous force, and pressure force are plotted vs. time. The Radial Piston Engine accelerate during 0.1 sec to nominal speed 500 RPM. It is clear that majority of the torque comes from the pressure forces, in that fluid forces acting normal to the surfaces of the 16 pistons and the cylinder block.

Particleworks is a CFD software based on an advanced numerical method known as the Moving Particle Simulation (MPS) method. The mesh-free nature of MPS allows for robust simulation of confined and free-surface flows at high resolutions, saving the time need to generate meshes for the fluid domain.
Particleworks is distributed in Europe by EnginSoft. Particleworks is a product of Prometech, a Japanese company, founded by experienced professionals and researchers at the University of Tokyo in 2004.

Bosch Rexroth Mellansel AB delivers complete hydraulic drive systems to several industry sectors, like Material handling, Marine and offshore, Recycling, Cement, Rubber, Sugar and different Mobile applications.

Daniel Svanbäck - HÄgglund Bosch Rexroth
Ragnar Skoglund – EnginSoft Nordic

For more information:
Susanne Glifberg - EnginSoft
s.glifberg@enginsoft.se

Energy and Automation

For over 90 years, LOVATO Electric has been designing and manufacturing low voltage electrical devices for industrial applications.
Founded in 1922 in Bergamo, Italy, LOVATO Electric is a privately owned company, and has been managed by the same family of entrepreneurs for 4 generations. Among the first Italian companies to achieve ISO 9001 certification in 1992, LOVATO Electric has a range of over 10,000 products that comply with the strictest requirements of international standards.
Motor protection circuit breakers, contactors, pushbuttons, switch disconnectors, limit switches, digital multimeters, energy meters, soft starters, AC motor drives, automatic power factor controllers and engine and generator controllers are just some of the products designed and built by LOVATO Electric. To provide competitive products and services in the industrial automation and energy management fields is our company's "mission". Our 13 branches abroad and a network of over 90 importers ensure the availability of Lovato Electric products in more than 100 countries all over the world.

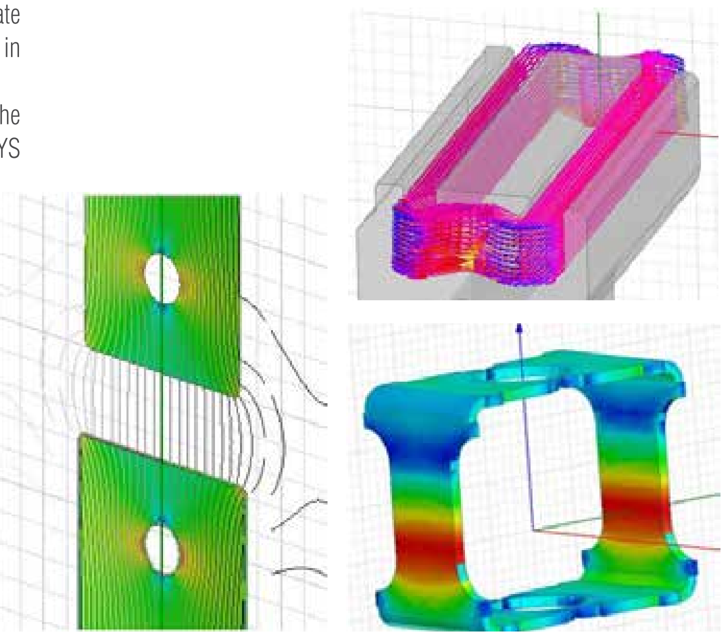
Why ANSYS Maxwell in Lovato

In LOVATO, the choice of ANSYS hasn't been driven by the pulse to turn to the world market leader, but has been the result of an accurate software selection by means of a pilot project verification carried out in collaboration with EnginSoft.
This project has been accomplished in two different phases. In the first one, a virtual model of a switcher has been created inside ANSYS Maxwell 3D, with the aim to reproduce its functioning and match the experimental data.
In the second phase LOVATO has exploited the parametrization and optimization tools inside ANSYS Maxwell to investigate different configurations and increase the overall switcher performances.
Eventually ANSYS Maxwell has proved to be the most accurate, advanced and easy to use tool among the valued ones. Nowadays ANSYS Maxwell is applied both in the first phase of the product design and to enhance existing products, guaranteeing that the project complies with the necessary performance requirements before its production starts.
ANSYS Maxwell is also used to both maximize the performance, for example in terms of magnetic forces and closing time of mobile parts, and increase the energetic efficiency of the magnetic devices.

"In Lovato we have always been in favour of all software solutions that can provide us with competitive advantage - declared Eng. Guaiatelli, CAE engineer in LovatoElectric - and in this perspective ANSYS Maxwell seems to be the right tool able to support us in the design of our products, allowing us to evaluate many design alternatives in less time, thus increasing their specific quality; in just few words to work towards INNOVATION, what everybody talks about and what we take very seriously into account."
"After a careful technical assessment, we have chosen ANSYS Maxwell since in our opinion it represents the best technology currently available on the market - continued Eng. Guaiatelli - moreover EnginSoft represents a serious and reliable partner who has assisted us with highly qualified professionals during the pilot project phase, enabling us to become quickly autonomous."

Eng. Iacopo Guaiatelli
CAE Engineer - Lovato Electric Spa

ANSYS Maxwell allows Lovato to reduce the number of prototypes which are necessary to guarantee the correct fulfilment of the regulation and to validate the project functionality at the same time. Therefore ANSYS Maxwell is capable to steer the design phase avoiding the risk of taking into account wrong strategies, lagging the products time to market.
In conclusion ANSYS simulation technology enables LOVATO to accurately predict how their products will thrive in the real world. LOVATO trusts ANSYS software to ensure their products integrity and to drive business success through innovation.



Production of Industrial Fans and Heat Exchanger since 1909

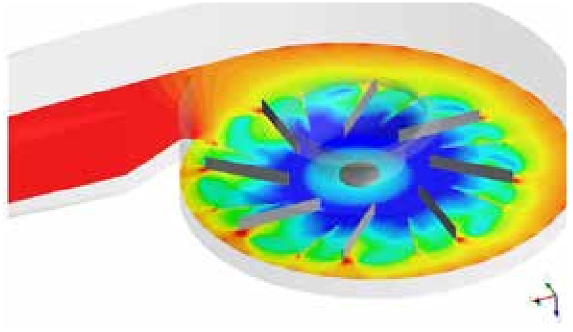
The Boldrocchi group is headquartered in Italy and has subsidiaries all over the world.
The company has more than a century of experience in offering specific solutions that combine various production lines.
Of the more than 500 employees worldwide, 260 work in production and 200 are engineers. The main factories in Biassono, near Milan, have a production area of over 35,000 m2 and are equipped with thermal, acoustic and aeraulic test facilities.
The Industrial Markets that Boldrocchi serve are:

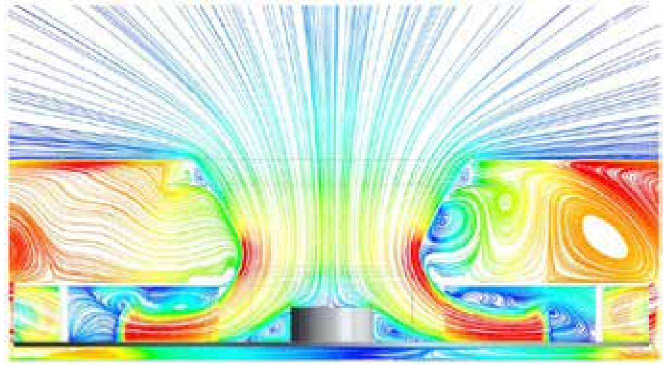
- Power Generation and Industrial Boilers
- Cogenerations plants & Gas Turbine
- Incinerations & Waste to energy
- Iron & Steel making industry
- Cement and Lime Industry
- Glass & non Ferrous Industry
- Oil & Gas, Petrochemical Industry
- Chemical, Process & Fertilizer industries
- Offshore, Marine and Shipbuilders
- Transformers, Generators and Motors

> "Boldrocchi has always focused on having the best available equipment and facilities, and we use the same philosophy when selecting our software solutions. After a careful assessment of the best available CFD simulation technologies, we have chosen ANSYS CFD, since in our opinion, it represents the best currently available software on the market that meets our needs" - Ing. Pietro Accurso, Compressor Office Manager of Boldrocchi.
> "The power of ANSYS CFD, together with its versatility and user-friendly approach, has convinced us that this software can provide a competitive advantage and lead to a strategic technical growth for Boldrocchi - continued Ing. Accurso - furthermore, EnginSoft has proven be a partner with great expertise in our industrial sector and will support us in the initial start-up phase, allowing us to be independent in the shortest time possible, thus leading us to a fast return of investment".
>
> Eng. Pietro Accurso
> Compressor Manager, Boldrocchi

ANSYS CFD in design

Our production in Boldrocchi is divided into five main lines: heat exchange (exchangers and coolers), fans, blowers, compressors, industrial noise protection, air pollution control/gas cleaning systems, GT intake/exhaust systems and heavy duty dampers.
The ANSYS CFD code is used by our turbo-machinery technicians of the FANS, BLOWERS and the new COMPRESSOR lines.
The CFD code previously used in Boldrocchi was not suitable to simulate high fidelity turbo-machinery CFD models as: transient conditions, real gases, advanced Turbulence and Moving Reference Frame algorithms (frozen rotor, stage and transient blade rows). All these features are available in ANSYS CFD.
Nowadays all the competitors in the turbo-machinery market rely on advanced CFD tools to design, optimize and verify their products. Low-level codes are not reliable, especially for compressible flows involved in compressors, while experimental data is too expensive in terms of time and costs. ANSYS CFD is used to analyze and





optimize the turbo-machinery in our industrial applications with the aim to increase the efficiency of both existing and new models of fans and compressors. ANSYS CFD allows us to reduce the number of prototypes necessary, to guarantee the customer requests in terms of performance.
ANSYS CFD was chosen by our company as the best-in-class in the turbo-machinery CFD simulation, together with the awareness that the EnginSoft team will support us efficiently thanks to a skilled competency in turbo-machinery, from design to analysis and optimization.

ExaNeSt H2020 Project at HiPEAC Computing System Week in Zagreb

HiPEAC stands for High Performance and Embedded Architecture and Compilation and it's the focus of a European Network, whose mission is to steer and increase the European research in the area of high-performance and embedded computing systems, and stimulate cooperation between academia, industry, computer architects and tool builders.
The HiPEAC network intensively works to foster research and knowledge transfer, organizing conferences and Computing System weeks on a yearly basis.
The HiPEAC conference aims at providing a high-quality forum for computer architects and compiler builders working in the field of high performance computer architecture and compilation for embedded systems, but is also open to general-purpose research which is becoming increasingly relevant to the embedded domain.
In this perspective the last Computing System Week, held in Zagreb on April 27th and 28th , was really pleased to host the project meeting of ExaNeSt (European Exascale System Interconnect and Storage), one of its related projects, which develops and prototypes solutions for next-generation Interconnection Networks, Storage and Cooling, as these have to evolve for the production of exascale-level supercomputers to become feasible. The project provided a relevant key-note speaker presentation by Martin Kersten (Centrum Wiskunde & Informatica – CWI - NL) concerning big database management systems, highly complex queries aimed at supporting a wide spectrum of market scenarios for the upcoming future.
This event in Zagreb provided a great opportunity for the ExaNeSt consortium for networking, disseminating some project activities and to start the preparation of the next project review meeting, due at the end of July. It will represent the first official milestone to show and confirm the great progress and achievements obtained by the project up to now, considering its high ambitions and the related expectations relying upon it.

CALTRA uses particle flow simulations to solve equipment problems

Consulting company invests in Rocky DEM software to detect failures in transportation and ore beneficiation processes, and to develop reliable solutions for customers

Maintenance stops, clogging, and particle flow problems are common in mining equipment. Identifying and correcting these issues are not simple exercises; it is necessary to have the right knowledge, technique, and tools. CALTRA – a consulting company specializing in industrial engineering and material handling equipment design – has found that DEM (Discrete Element Modeling) simulations are the best way to develop efficient solutions for their customers. The CALTRA engineering team uses Rocky DEM software to ensure the reliability and quality of their customer's projects.

"Rocky DEM allowed us to evaluate equipment with higher precision. With the introduction of particle parameters that were closer to reality, as well as the possibility of simulating with larger material volumes, we were able to bring clarity to our data. If we hadn't used Rocky DEM, those analyses would be limited," says Bernardo Augusto Roca Arenales, General Manager at CALTRA.

Fig. 1 - CALTRA ensures the reliability and quality of their customers' projects with simulation software

Chute Clogging

CALTRA has more than ten years of experience in equipment development and, recently, analyzed several chutes from an ore mining plant, eight of which were found to be clogging during the ore pass.

To increase the equipment productivity, the mining maintenance team had to operate with scheduled shutdowns to unclog the chutes and mix water into the material. The inclusion of water to the ore changes the material properties and flow of the ore as well as speeds up the equipment oxidation.

To solve these problems, the CALTRA engineering team used Rocky DEM to analyze the material flow and identify blockade points in the equipment. Their redesign goals were to avoid significant changes to the structure of mining plant, which was impractical economically. Thiago Rodrigues Ferreira, Mechanical Engineer at CALTRA, said that based on Rocky DEM results, some low-cost actions were suggested to the customer in order to combat and minimize problems.
"We analyzed the chute, then designed and tested a solution to eliminate the clogging. We also equalized particle and belt speeds, because the closer and more similar they are in direction and velocity, the smaller the impact is to the equipment. In addition, we centralized the material flow over the belt to reduce misalignment," explains Ferreira. "The actions we proposed ensures that the chute will not clog again," he says.

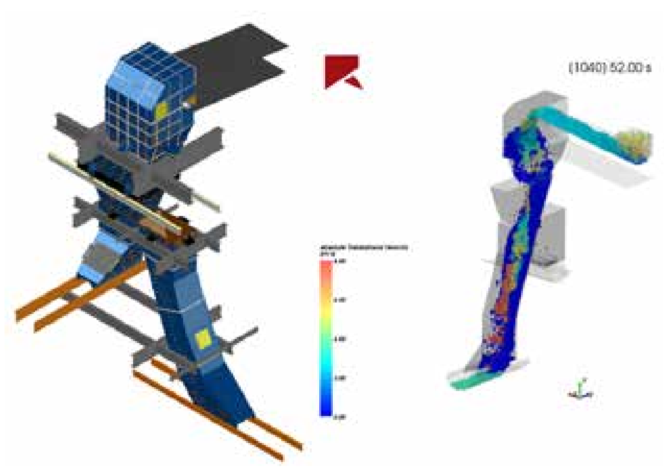


Fig. 2 - With the help of Rocky DEM software, CALTRA identifies areas of clogging and proposes design changes to improve chute performance

Presenting the solution to the customer

To present the redesign and demonstrate the overall quality of their solution, CALTRA once again used Rocky DEM, and especially the simulation video resources to show the customer different perspectives and angles of the chute with 3D views and movement.
"We used many images and videos to present a general panorama to the customer, comparing different solutions," says Bernardo Augusto Roca Arenales, General Manager at CALTRA.

To the CALTRA team, the use of DEM software brings benefits both to the development of projects and to the presentation of the results. "What is interesting about Rocky DEM is that it enables us to conduct specialized technical analysis during the problem solution phase, which helps us identify the solution faster. This saves our engineers time and enables them to focus more upon theories and technical issues," says Arenales. Besides this, he says, the solutions obtained with the Rocky DEM software helps CALTRA attract new customers. Prospects who demand high quality and reliable service are impressed with the features the technology provides.

Rocky DEM benefits for CALTRA

CALTRA bought the software in the second quarter of 2015 and has already discovered many benefits of using the tool. One of the main aspects highlighted by Ferreira is the ability of Rocky DEM to represent accurately and realistically the equipment and material, while allowing them to work with large volumes of particles.
To Ferreira, the benefit of Rocky DEM when compared with other DEM softwares is that it allows them to create different shapes of particles. Generally, DEM software allows the generation of only rounded particles, and to analyze a non-spherical particle it is necessary to join two or more round particles to obtain a non-regular geometry (sphere clumps). Thus, the ore is simulated with two, three, or more particles. Rocky DEM allows users to create irregular geometry particles - polyhedrons, polygons, multifaceted, among others - resulting in just one particle. This allows engineers to simulate much larger volumes of material without exceeding hardware limits.

Challenge

Propose changes to several chute designs that reduces maintenance stop time and avoids clogging, without significantly modifying the mining plant structure.

Solution

By simulating the chutes and flow in Rocky DEM and matching the material composition, the engineers were able to determine that by inserting self-aligning belts and matching the belt speed to the particle fall speed, they could accomplish their goals without changing the structure and distribution of the plant equipment.
Benefits
Rocky DEM simulations enabled CALTRA to propose to their client minor, low-cost modifications to the equipment with reliable results, reducing maintenance stop times and increasing the plant's productivity.

Testimonial

For CALTRA, Rocky DEM has fundamental importance because it allows us to evaluate with accuracy the equipment in the field. With this, we can represent reality and bring clarity of information by inserting particle parameters, moving elements into the simulation, as well as working with large material volumes. If we did not use Rocky DEM, our analysis would be much more limited.

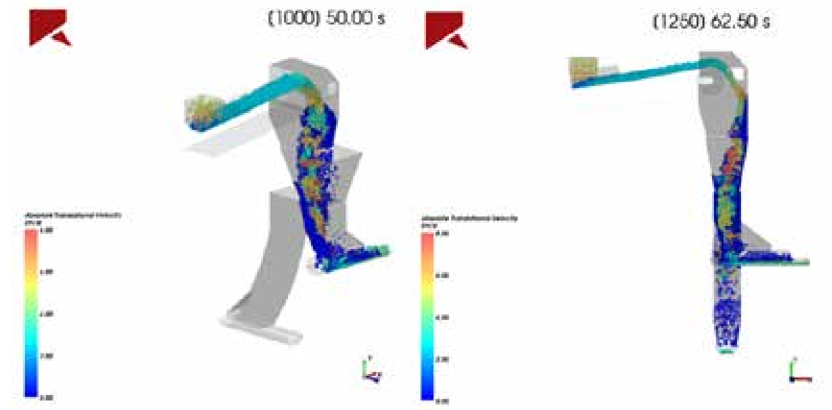


Fig. 3 - Rocky DEM's realistic simulation views and video animations helps CALTRA better demonstrate proposed solutions to their customers

CALTRA®

CALTRA

CALTRA provides consulting services for mechanical and structural industrial equipment, and designs material handling equipment. The company, founded in 2003, has expertise in solving several critical situations and is constantly seeking to expand its practical and theoretical knowledge and upgrading its management methods.

For more information:
Massimo Tomasi - m.tomasi@enginsoft.com

Reliability-based Robust Design Optimization Using Polynomial Chaos Expansion for Aeronautics Applications

This article describes the methodologies developed by ESTECO during the UMRIDA European Project, dedicated to Uncertainty Management and Robust Design Optimization and implemented in modeFRONTIER software platform.

As proved by recent studies [Clarich et al., 2009], the application of Polynomial Chaos expansion is one of the most efficient methodologies to accurately manage uncertainties in industrial design [Loeven et al., 2007]. However, this methodology requires a minimum number of samples, which heavily increases in proportion with the number of uncertainties and a typical industrial optimization case (for instance, at least 10 simultaneous uncertainties) can hardly be treated as a feasible task.

For this reason, we propose some innovative approaches to efficiently handle industrial problems of this kind, both in terms of Uncertainty Quantification (UQ) and in terms of Robust Design Optimization (RDO).

For the UQ, we propose the use of methodologies that allow to identify which Polynomial Chaos terms have higher statistical effects on the performances of the system, reducing the number of unknown coefficients and therefore the number of needed samples to complete the UQ, without necessarily discarding an uncertain variable from the problem (keeping this way a higher accuracy).

For the RDO, we propose a methodology based on the reliability formulation of objectives, which guarantees the reduction of objectives number compared to a classical RDO approach [Kalsi et al., 2001] and therefore the possibility of drastically reducing the number of configurations to be evaluated and simulations to be performed. In order to guarantee an accurate application of this methodology, we developed an approach based on the exploitation of Polynomial Chaos coefficients to accurately evaluate the percentiles of the quantities to be optimized/constrained. This methodology is also called reliability-based design optimization [Clarich et al., 2014].

The solution based on Polynomial Chaos exploitation is innovative and very promising in terms of efficiency. Its application to a case of industrial interest, proposed by Leonardo Aircraft, confirms

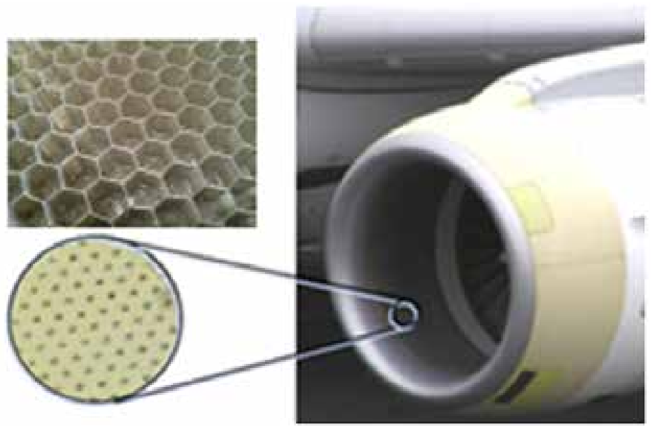


Figure 1 - Full engine nacelle (right) and detailed views of the acoustic panel interior (upper left) and perforated facing skin (lower left)

the validity of this methodology. In this specific case, a nacelle acoustic liner of typical regional jet engine inlet, subjected to manufacturing tolerances, is optimized to reduce the overall noise emissions in three certification conditions and to decrease the bandwidth on performance distribution.

Definition of Robust Design application

The industrial validation case targeted in this study is a typical regional jet engine nacelle integrating an acoustic liner consisting of a number N of Helmholtz resonating cavities. Each one is essentially composed of a honeycomb cell of a specific height targeted to attenuate the design frequency of the impinging noise, being closed at the bottom by a rigid backing skin and on top by a porous face-sheet to dissipate energy as the air passes through.

Acoustic requirements are prescribed by customers and aviation agencies for three typical flight conditions: Approach; Sideline (or Take-Off); and Flyover (or Cut-Back)

The geometrical uncertain parameters considered are: Honeycomb height (h); Face sheet thickness (tfs); Face sheet open area (PoAeff); Holes diameter (deff) [Copiello et al., 2009].

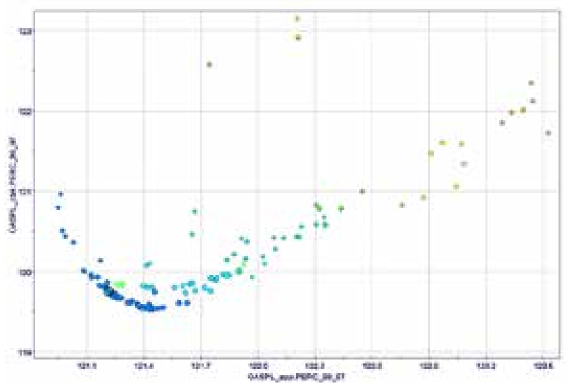


Figure 2 - Objective function (OASPL 99.97 %-ile) for each flight condition (abscissa, ordinate, colour), per every proposed design

These geometrical parameters are used by a proprietary semi-empiric impedance model [Murray et al., 2005] together with values depending on flight condition, as input parameters to compute the admittance data, necessary to specify the boundary conditions of the FEM model of the acoustic liner. The acoustic numerical simulations have been performed by MSC-ACTRAN software, on an 2D axisymmetric model consisting of about 30,000 quadratic elements. As objective function it has been considered the Over All Sound Pressure Level (OASPL), which uniformly sums over the directivity angles and over frequencies the Sound Pressure Level (SPL) which is a function of the frequency, of the duct radial and azimuthal modes root mean square acoustic pressure, and of directivity angle at a distance of 150 ft.

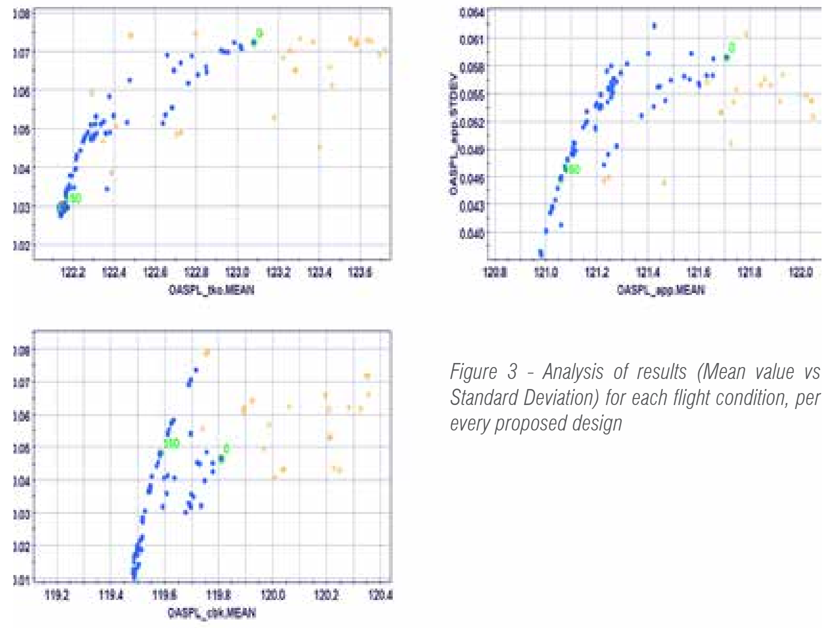


Figure 3 - Analysis of results (Mean value vs Standard Deviation) for each flight condition, per every proposed design

Optimization results

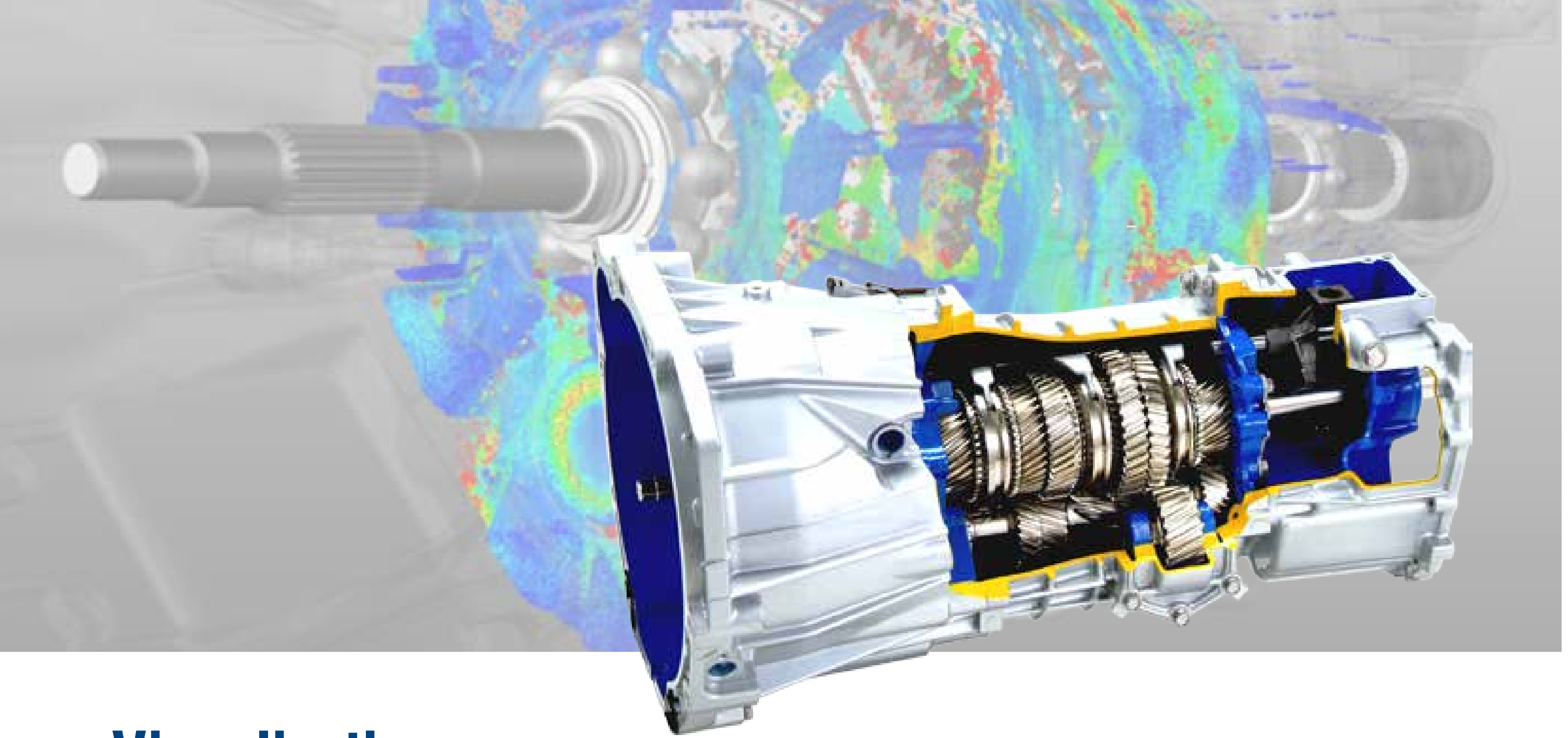
A campaign of experiments has been conducted by Leonardo Aircraft, which has performed the geometrical measurements of a series of acoustic panels. The database was therefore been imported in modeFRONTIER software from ESTECO, and analyzed with the Distribution Fitting tool in order to find the statistical distributions which better fit the experimental data; the properties of the statistical distribution can then be used in the optimization problem to define the stochastic parameters for each design proposed by the optimization. Once the geometrical parameters uncertainties have been quantified, it has been possible to integrated the numerical models in the optimization software modeFRONTIER of ESTECO, which allows the automatic variation of the parameters and the performance evaluation for every configuration proposed by the selected optimization algorithm, until the optimal solution is found. As objective function, it has been selected the 99.97 %-ile value of the OASPL function distribution, summed up for each of the three flight conditions considered, and obtained by the sampling of the uncertain parameters distribution: thanks to the methodology based on PCE (Polynomial Chaos Expansion) available in modeFRONTIER, it has been possible to reduce the sampling size to 15 simulations only for each design.

By the evaluation of about 150 different designs (figure 2), it has been possible to select an optimal configuration which reduce the 99.97 %-ile of the OASPL signal distribution of around 0.7 dB for each flight condition.

Acknowledgments

This project has received funding from the European Union's Seventh Framework Programme for research, technological development and demonstration under grant agreement no ACP3-GA-2013-605036. The authors wish to thank Nicola Magnino from Leonardo Aircraft for the collaboration with ESTECO during the project.

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



Visualization of Oil Lubrication in the Transfer Case and the Transmission using Particleworks

Oil has various important roles in lubrication, cooling, buffering and air tightness, whereas it causes torque loss for its flow resistance

Univance Corporation is a vertically integrated company which specializes in the development, manufacture and assembly of automotive parts and units. They have been pursuing their unique technologies based on business' units including all wheele drive, automotive parts,gearbox. Now they deal with functional components such as one-way clutches, AT/CVT components, transmission and engine gears, EV/HEV gearboxes, manual transmissions, and 4WD transfers cases (Fig1).

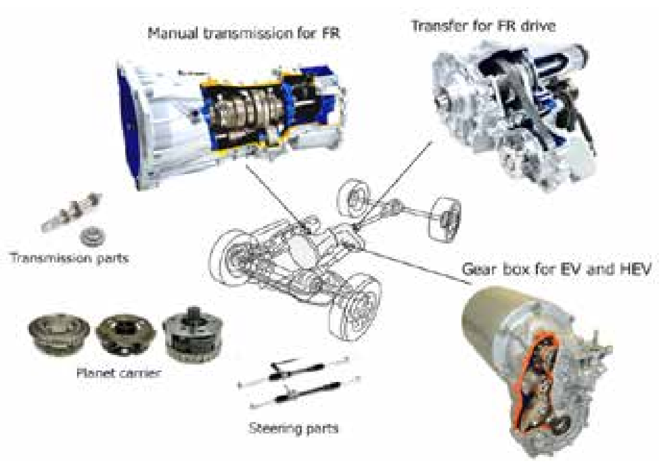


Fig. 1 – Products for automotive

In the product development department, they use CAD and CAE to support design process of all products. They generally use CATIA V5 as their standard CAD system and perform simulation by CAE tools including Particleworks by importing CAD models created in CATIA V5. The purposes of the simulation are, for example, prediction during the concept design stage, design validation before prototype testing, and analysis for quality improvement. Especially in the CAEgroup in charge of further analysis, a wide variety of simulation, such as linear-structural analysis, Particleworks for fluid analysis and RecurDyn for MBD (Multi Body Dynamics) coupling with Particleworks, is used as CAE.

Introduction of Particlesworks into Univance Corporation

Oil has various important roles in lubrication, cooling, buffering and air tightness, whereas it causes torque loss for its flow resistance. For that reason, enough considerations and arguments are necessary for deciding conditions of oil physical property and quantity, and shape optimization. Therefore there was a requirement in Univance Corporation to confirm the real phenomena by simulation because it was too difficult to understand it only by experiment, which couldn't be realized at that time. After a while, the request to visualize oil sloshing and lubrication for sufficient evaluation has increased and they've finally come into introduction of Particleworks through comparison with other CFD competitive solvers and benchmark test. In addition to fluid analysis, they need co-simulation of the chain behavior and the oil lubrication in the transfer case which is their key product. In other words, MBD has been a must and RecurDyn which can be coupled simulation with Particleworks has been introduced as well.

Simulation Example 1 of Particleworks: Visualization of Oil Lubrication in the Transfer Case

Univance Corporation gives a simulation example of oil lubrication in the transfer case. The transfer case is a device to distribute power from engine-transmission to front wheels and rear wheels of a 4WD vehicle by means of gears, sprockets and a chain (Fig2). They used RecurDyn and Particleworks to simulate the mating of the chain and the sprocket and the oil lubrication. Fig3 shows full model of the chain and sprocket mating, and enlarged images focusing start and end of the chain and sprocket mating are shown fig4 and fig5.

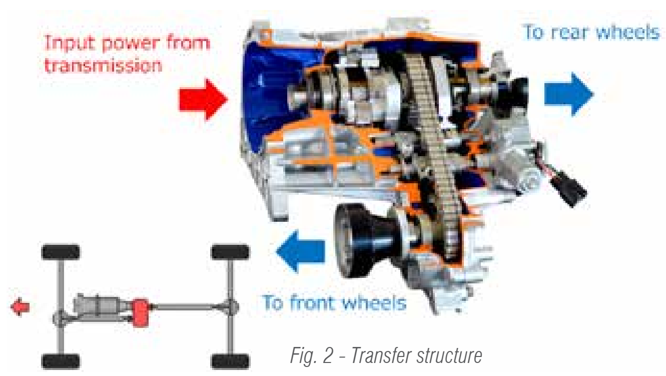


Fig. 2 - Transfer structure

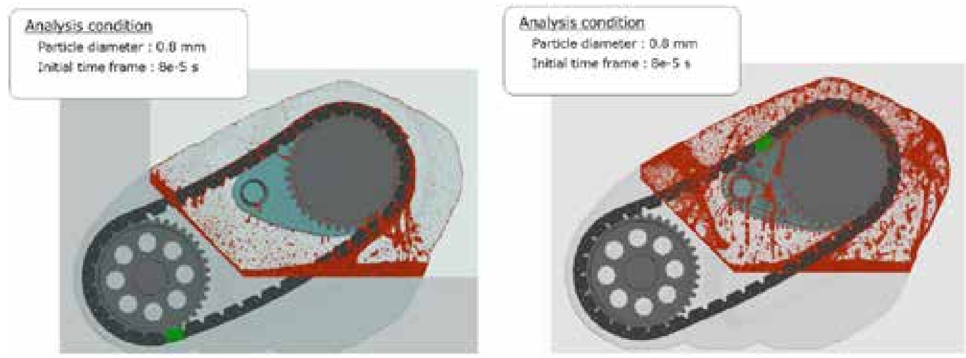


Fig. 3 - Full model of the chain and sprocket mating (left:75rpm, right:400rpm)

Before, they had visually checked the oil lubrication created by the chain using resin clear case and endoscope. Such visual inspections have limitation, and it's difficult to take measures because the inspection is after making real products and significant layout change is impossible even if there are big problems. However, oil lubrication is necessary for antifriction and cooling of the components and has the important role to improve the product quality. It became possible to be visualized by simulation to evaluate how the oil circulates and how the oil flows after circulation.

Regarding to the practical issue of calculation cost, the simulation using Particleworks needs only one day at most for calculation thanks to great improvement of the software and hardware-development such as GPU. The number of particles that Particleworks can handle in simulation largely increased, which let us use a smaller size of particle (diameter) compared to before and perform more accurate simulations.

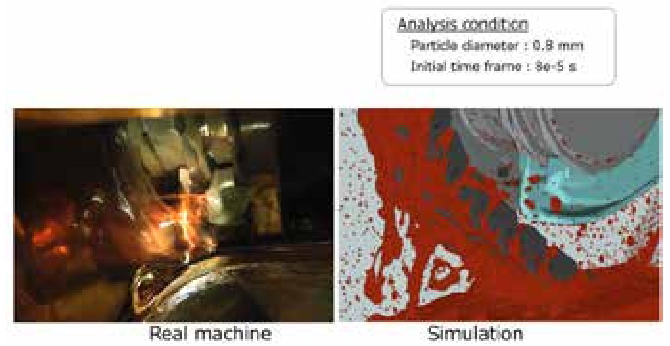


Fig. 4 - Start of the chain and sprocket mating (75rpm)

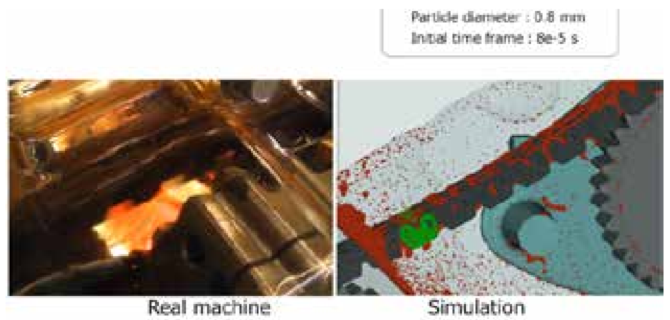


Fig. 5 - End of the chain and sprocket mating (75rpm)

Simulation Example 2 of Particleworks: Visualization of Oil Lubrication in the Transmission

Nextly, a simulation example of oil lubrication in the transmission is introduced below. The transmission is a machine in a power system, which supplies controlled application of the power. The word "transmission" can be referred as the gearbox consisting of gears and gear trains to provide speed and torque conversions from a rotating power source into other devices. Little oil isn't enough to circulate around the whole gear box and not to work properly its functions of lubrication and cooling, and has a cause of gears' burning. However too much oil gives an extra torque to gears and shafts. Therefore, if we could confirm the oil behavior in the gearbox, we would be able to decide suitable shape and necessary quantity of oil. As mentioned before, visualizing the real phenomena by experiment is difficult, in that case, Particleworks comes into effect to understand the oil behavior. Unlike

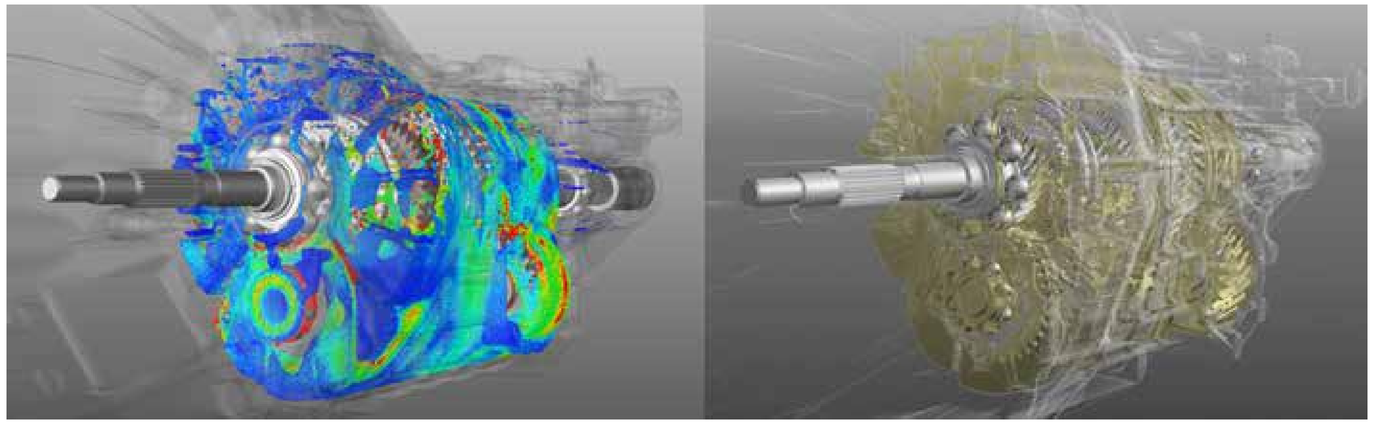
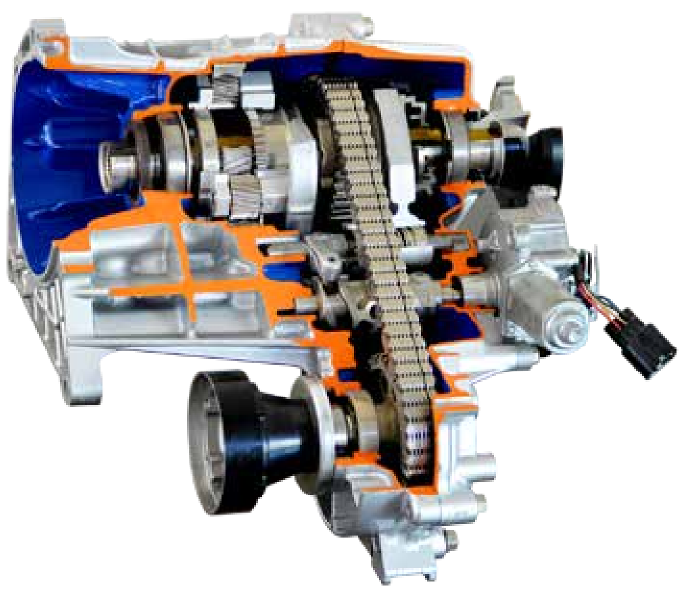


Fig. 6 - Full model of the transmission (left: particle image, right: post-rendering)

Example 1:Oil Lubrication in the Transfer Case, this simulation is conducted only through Particleworks. Besides, each gears is given two rotational patterns: low-rotation and high-rotation.
The visual images of the result are given above (Fig6). The left of Fig6 shows the result in particle and the right is rendered image. Using Particleworks, which strength is to easy to analyze free surface flow and large deformation, makes it visualize where and how the oil flows and circulates, and can simulate under the wide variety of conditions by changing not only rotation pattern like this example but also the shape of model and physical property of oil. Moreover, we can evaluate the velocity of oil and the quantity of oil circulating around the model, with post-processing capability. Fig7 shows the graph of torque with changing gears' rotational speed to quantitatively estimate the result of simulation. According to the graph, it is confirmed that the torque increases corresponding to incensement of rotational speed, which the same trend can be seen in the experiment.

However, the values gained through experiment are the total values included of various factors and Particleworks deals with only torque loss due to fluid resistance without considering frictions among gears, bearings, and collusions of each parts. Therefore, co-simulation with RecurDyn for oil lubrication in the transmission is needed for reproduction of the phenomenon above and better development.



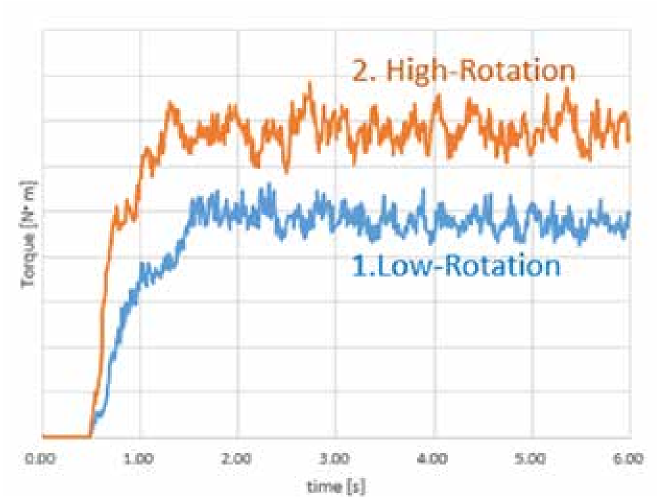


Fig. 7 - Torque graph with changing rotational speed

Conclusions

In this paper, we introduce our Particleworks and RecurDyn user Univance Corporation with their simulation examples. Oil plays an important role in lubrication and cooling of components and can be a cause of torque loss at the same time. Experiment to reproduce the oil behavior is technically and physically difficult problem and fluid simulation such as Particleworks comes into solution. Simulation of oil lubrication in the transfer case and the transmission shows the result that strong-coupling Particleworks and RecurDyn method can simulate not only fluid itself but also interaction between the fluid and components.

Reference

Particleworks users interview UNIVANCE CORPORATION
T. Morimi and N. Fujita, "Analysis and Evaluation of Stirred Fluid Flow Resistance Using Moving Particle Semi-implicit (MPS) Method", Proceedings of Computational Engineering Conference Vol. 21, 2016 May.

Toshiyuki Morimi, Prometech Software Inc.,
For Particleworks inquiries: info@enginsoft.it

Optimum Design of Diesel Ship Engine Silencer

It is of great importance to properly design and create a supporting construction for diesel engince machines, the main source of underwater noise on ships and which create airborne sound, exhaust system and system equipment all of which are designed in accordance with the ship on which they shall be mounted. For that reason in order to increase domestic capital proportion in battleship projects and also in order to serve as an example to Turkish Defense Industry; facilitating design and manufacturing processes of the abovementioned exhaust systems and its related equipment inside Turkey, which is something the domestic industry could not achieve yet, were being demonstrated in this Project by making a sample exhaust system design for a battleship Project and silencers included in this exhaust system were physically manufactured. Through acoustic measurements obtained en-route, it was observed that the system displayed the expected requirements of en-route performance. The study was completed after utilization and implementation of the optimization method.

1. Introduction

Main propulsion systems in new civil and military ships shall be generally based on diesel machine engines, also taking into consideration sizes and purpose of use of ships in construction. Use of diesel machine engines increase the importance of both design and post-design technical features of the diesel engine machine and related equipment.

In Turkey, exhaust system and exhaust system equipment of diesel engine machines for battleships are procured from overseas vendors either imported in packed form or directly procured through overseas exhaust system manufacturers. During this project, in order to increase domestic capital proportion in battleship projects and also in order to serve as an example to Turkish Defense Industry; facilitating design and manufacturing processes of the abovementioned exhaust systems and its related equipment inside Turkey, which is something the domestic industry could not achieve yet, were being demonstrated in this Project by making a sample exhaust system design for a battleship project and silencers included in this exhaust system were physically manufactured.

2. Exhaust System Back-Pressure and Thermal Analysis

Back-pressure value displayed upon back-pressure analyses conducted on all exhaust systems were compared with back-pressure criteria designated by the manufacturing company and the dimensions of silencer construction mounted on the exhaust system as well as the exhaust system were determined while also taking into consideration the acoustic characteristics of the silencer and back-pressure criteria values.

2.1. Silencer Model Back-Pressure Analysis

Under the scope of exhaust system design, a silencer model [1] compatible with the layout of machine and which includes a perforated tube and reflection chamber designed to silence the noise was developed.

Using the environment and limit conditions determined for the analysis, average back-pressure value on the silencer was measured at 6.370 Pa (63,70 mbar). Back-pressure value obtained from the silencer is over the value provided for the silencer by the manufacturing company (between 30-50 mbar) and it was determined that this silencer construction shall not be deemed valid and that internal structure and dimensions of the silencer must be altered.

For the abovementioned silencer, due to the fact that exhaust gas input and output were made through the same hole, which causes the exhaust gas to travel the same distance to and from and that cross-sectional area of the input circuit having a small area which causes the exhaust gas flow rate to be high,and that exhaust gas making a sharp turn once it completes the way and turns towards the output circuit which causes an increase of flow rate and due to losses caused by turbulence flow; back-pressure values obtained from this silencer was higher than the criteria set.

2.2. Improved Silencer Model Back-Pressure Analysis

Under the scope of exhaust system design, design of the silencer (dimensions and internal structure) which will be used in the system was created taking into consideration acoustic design and engine room layout, details of which will be explained later in this study. Taking into consideration diesel engine exhaust output dimensions; the circuit

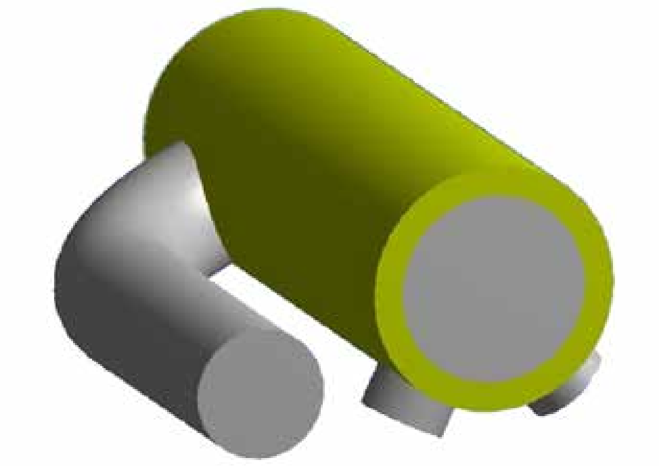


Figure 1 - Improved Silencer Design

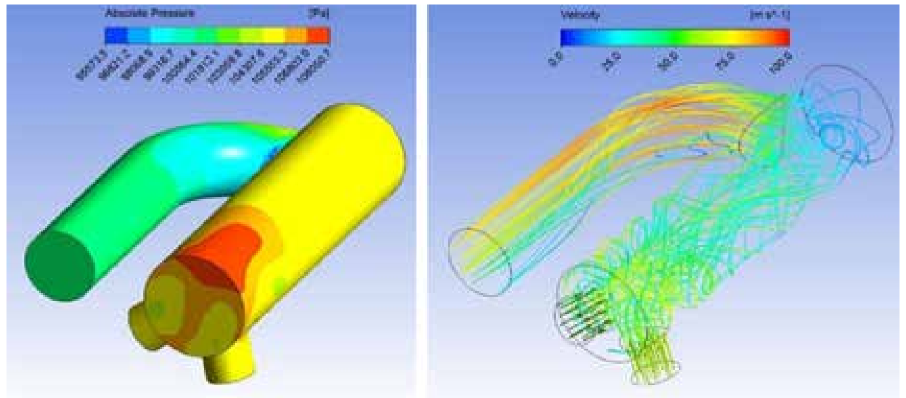


Figure 2 - Improved Silencer Flow Lines

which is expected to be located at the exhaust input were designed in two alternatives and among these two alternative geometries, the one with the lowest pressure loss was selected as the (primary) silencer design.

As a result of pressure loss analysis using the new geometry, back-pressure values were reduced to 2.300 Pa (23,00 mbar). Considering the back-pressure value results obtained, it was determined to approve the second geometric design of the silencer as the final design and to use this design in the exhaust system.

3. Thermal Analysis of Isolation Material

Exhaust gas has a high temperature and in order to reduce this temperature in order to prevent the personnel who will work inside the engine room from being exposed to high temperature, it was determined to coat the whole exhaust system with isolation material. While determining the thickness of isolation material, due to special conditions of the engine room (when compared to rest of the exhaust system), a thermal analysis was performed taking into consideration the thickness of isolation material used on the silencer. While performing thermal anaylses, maximum exhaust temperature (430 °C) and heat conductance coefficient of steel profile material and isolation material were taken into consideration.

As a result of the thermal analysis performed, surface temperature on the surface of the isolation material was measured as 77,6 °C, under minimum isolation material thickness. Considering the fact that this minimum thickness of isolation material will increase in other parts of the exhaust system, it was determined that the temperature value obtained as a result of thermal analysis was applicable for engine room conditions.

4. Acoustic Design of Exhaust System

Silencers are noise control tools which are used to reduce the noise created by diesel engine exhaust, fan and other noise sources in areas with gas flow. Under the scope of exhaust system design and in order to create a design for the silencer which will be used on the system; a silencer model which has a perforated pipe inside the structure (inner) and which has reflective feature however essentially reducing the noise with its swallowing feature gained via the isolation material (stone wool) coated around the perforated pipe and which is also compatible with the layout of the engine room. Silencers which has swallowing feature can be designed either in rectangular tube or circular cross-section form. Design methodology used was the same for both however silencers with circular cross-section reduces noise level two times more than rectangular tube models which are at same sizes, dimensions and which use the same amount of isolation material. For that reason, design of the silencer was determined to be in circular cross-section form. Noise characteristics of the diesel engine (Sound Pressure Level, SPL) which will be created at central frequencies of 1/1 octave bands were provided by the manufacturer and these central frequencies served as a benchmark for noise reduction amount. Under the scope of the design methodology applied on the silencer, sound speed (540 m/sn) of the exhaust gas under maximum temperature (430 °C) of the diesel engine were taken into consideration at first and wave length was determined for each central frequency at 1/1 octave band and Mach coefficient (rate of exhaust gas speed against sound speed) was also determined. Reduction amount corresponding to the unit length of the silencer was found using these parameters and after applying necessary fixes, noise reduction

Total SBD (dBA) - Given	130,30
Total SBD (dBA) - Reduced	99,60
Noise reduction (dBA)	30,70

Chart 1 - Silencer Acoustic Analysis

values based on its dimensions and internal structure were calculated taking into consideration the structure of the intake and exhaust gas dilation inside the intake. A-weight fix, which is used for frequency and sound pressure sensitivity of human ear, and range fixes were applied on these reduction values while taking into consideration that these measurements will be made from 1 meter distance, hence acquiring final reduction value (SPL) for each central frequency. SPL values, which were received by the diesel engine manufacturer and values obtained from the acoustic analysis for each central frequency were summed up using logarithmic sum method, hence Total Sound Pressure value was calculated and frequency-independent noise reduction amount for the silencer was determined.

Noise reduction level provided above are estimated reduction amounts using the silencer and they are considered sufficient for an exhaust system.

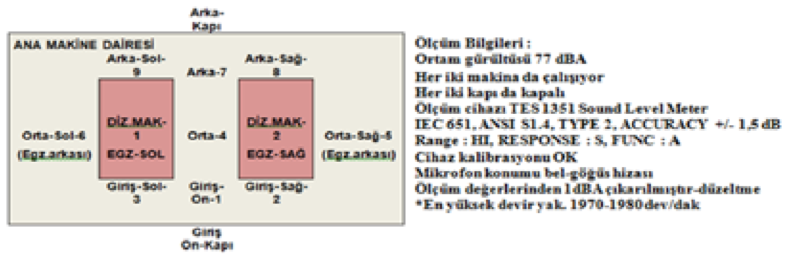


Figure 3. Main engine room measurement points and measurement information

Mot.Dev. (dev/dak)	Giriş-Ön 1	Giriş-Sağ 2	Giriş-Sol 3	Orta 4	Orta-Sağ 5	Orta-Sağ 6	Arka 7	Arka-Sağ 8	Arka-Sol 9
500	97	96	98	101	100	99	97	95	95
700	101	100	101	104	105	103	100	99	98
900	102	102	103	105	106	105	101	99	99
1000	101	103	102	105	106	103	100	99	98
1300	101	102	102	104	105	103	100	100	99
1500	103	103	103	106	107	104	102	101	100
1700	106	106	107	109	109	110	104	103	103
1900*	108	107	109	111	113	114	107	104	104

Chart 2 - LA values measured during different rolls/cycles of the engine (dBA)

5. Noise Measurement Results

The silencer was manufactured upon completion of its design and the whole exhaust system was mounted/assembled in compatibility with the engine room of the ship. In this study, under the scope of real environment verification of acoustic characteristic values obtained, several tests were performed.to measure en-route values. LAmax values measured during different rolls/cycles of the engine, which are defined under respective NATO standards are displayed on Chart 1. Higher limit at 1300 rpm under same standards are set as 110 dBA.

When nine different spots at 1300 rpm were measured, maximum dBA was recorded at 105 dBA. This value is 5 dBA below the maximum limit defined under the respective standard. As a result, it was proven via measurements that the designed silencer have expected acoustic characteristics.

As a result of back-pressure measurements performed on the ship, it was understood that measured values were below the pre-defined criteria.

6. Silencer Optimization Process

modeFRONTIER software was used for automatization of the optimization process and for using different algorithms at the same time. The method explained in this section was transferred to Excel macro for that purpose and was used in optimization process. At the same time, parametric values were added to ANSYS Workbench software and this allowed modeFRONTIER software to alter these values. Work flow diagrams used in modeFRONTIER software are displayed in Figure 6.

Acoustic analysis inputs are as follows:

- Sound pressure level [dBA]
- Frequency [Hz]
- Flow rate
  - Internal silencer radius (2h)
  - Isolation thickness (I)
  - Silencer length (L)

Among these inputs: sound pressure level, frequency and flow rate are provided by the manufacturer and are included in the process as fixed values. Silencer internal Radius, isolation material thickness and silencer length are also included in the acoustic measurement process and have pre-determined intervals. Starred values (*) displayed under interval charts represent variable values in the benchmark/reference design. Acoustic analysis input total noise maximum limit was set as 98 dBA. The purpose of this research is introduced to the platform as "minimization of noise level" and how this purpose effects the whole optimization process is observed directly.

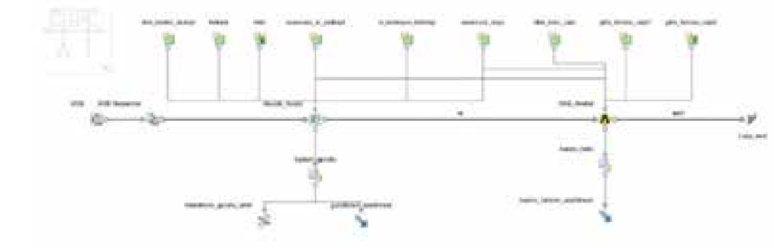


Figure 4 - modeFRONTIER work flow diagram

Process continues with calculated liquid dynamics analysis, which are performed after acoustic analyses. Inputs in this discipline are as follows:

- Flow rate
- Silencer internal radius (2h)
- Silencer length (L)
- Output pipe radius
- 1. and 2. İnput pipe radius

*Frequency [Hz]	*Sound Pressure Level [dBA]	*Flow rate [m³/s]
31,5	99	8,69
63	113	
125	121	
250	127	
500	123	
1000	122	
2000	118	
4000	111	

Chart 3 - Fixed acoustic analysis entries based on reference design

Internal Radius interval of the silencer [m]	Silencer length interval [m]	Isolation thickness interval [m]
(0,55 – 0,75)	(2,3 – 2,7)	(0,1 – 0,25)
*0,65	*2,5	*0,15

Chart 4 - Fixed acoustic analysis entries based on reference design

Silencer internal Radius interval [m]	Silencer length interval [m]	Output pipe radius interval [m]	1. ve 2. input pipe Radius interval [m]
(0,55 – 0,75)	(2,3 – 2,7)	(0,1 – 0,25)	(0,275 – 0,325)
*0,65	*2,5	*0,15	*0,3

Chart 5- Variable HAD Analysis Input

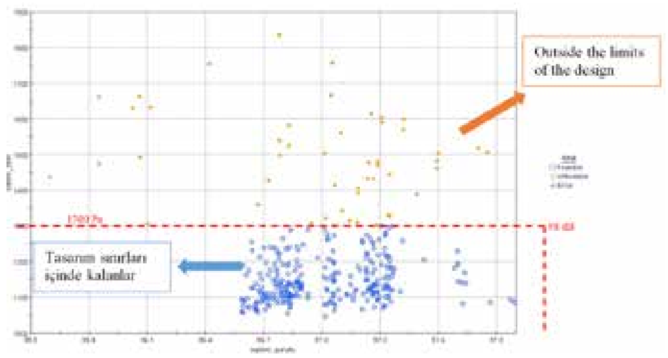


Figure 5 - Design space based on maximum total noise and maximum pressure difference

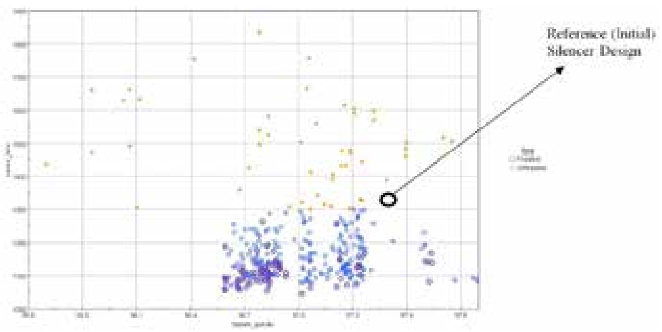


Figure 6 - Demonstration of optimum design cluster on global space

Other than flow rate, all input values used in calculated liquid dynamics analysis are defined between a certain interval. Among these variables; flow rate, silencer internal Radius and silencer length are identical to variables used in acoustic analysis.

Calculated liquid dynamics analysis are conducted based on the benchmark/reference design and pre-determined intervals. HAD analysis output total pressure difference (back-pressure) maximum pressure difference was set as 1300 Pa. The purpose of this calculation is introduced to the platform as a means of minimizing the total pressure difference.
In this study, MOGA-II (Multi Objective Genetic Algorithm) algorithm was determined to be applicable for multi purpose optimization process and it has been believed that this brings out better results within shorter time frames. Design methodology, automation of which was maintained, performs analyses one after another (consecutively)

7. Optimization Results

Results obtained in this study, which uses the design methodology in which the whole process is automatized are satisfactory. Under the scope of this study, 437 different types of designs were created and compatibility of each of these designs to pre-determined purposes and criteria were detected automatically.

As it is displayed on Figure 7, while some of 437 different designs have a noise level below 98 dBA and have a total pressure difference below 1300 Pa whereas some are beyond these limits. Limits compatible with designer expectations and limits that determine which designs are better are within the limits designated on the figure.

Figure 8 display the territories in which benchmark/reference design is located and also display how satisfactory the design is before the methodology was created. Reminding the audience that brown boxes represent designs outside predetermined limits whereas blue circles represent designs within predetermined limits; blue circles with red marks indicate optimum design cluster. As it is explained mathematically, in multi purpose optimization problems there is no such thing as one and only best design. If there are more than one purposes, there is a cluster called Pareto, which defines the cluster of best designs. Making the best selection among this cluster becomes a matter of choice fort he designer.

8. Results and Evaluation

A silencer is designed in this study. This design has been confirmed with analyses and measurements. The method developed for silencer acoustic design was used successfully in this study. In the last stage of the study, acoustic design and liquid analysis methods were automated with the help of modeFRONTIER software, which is an optimization platform, and then undergone optimization.
During the process, settling liquid analyses of optimization algorithms in a way to reduce the pressure difference worked out and average pressure difference was reduced to 1100 Pa levels. Likewise, noise level of benchmark/reference design was reduced to 96.7 dBA levels towards the end of the process.

Şadi KOPUZ, Hasan AVŞAR, Nasser GHASSEMBAGLOU - EnginSoft
Levent KAVURMACIOĞLU - İstanbul Teknik Üniversitesi

For more information: Şadi Kopuz - EnginSoft
s.kopuz@enginsoft.com

PASS – Piping Analysis and Sizing Software
Powerful Piping Simulation Tool for Every Engineer

Pipelines are everywhere. Known from times of ancient Rome and even earlier (Bronze Age Indus Valley Civilization by 2700 B.C.), they now became "blood circulatory system" of modern civilization. They supply hot and cold water and natural gas to your home, transport wastewater, provide you with heat and ventilation. They work in your car suppling fuel for the engine, and moving out exhaust gas, circulating coolant and lubricant. They transport oil and gas across countries and continents from onshore or offshore wells to refining and petrochemical plants. They provide fuel and circulate steam/water at power stations to give you power. They link equipment in process plants providing work of refining and chemical, metallurgy and mineral processing, biotechnological, food and water treatment facilities, helping to provide you with modern materials and products for your life. They connect tanks and vessels at storage facilities and gas stations, delivering fuel for road transport, ships and airplanes. And they are moving us into space transporting propellant and oxidant into jet engines of rockets and spaceships.
So it is hard to overestimate how important is to provide effective and safe work of piping systems. Piping transportation consumes about 20% of all world electric power. And what is more important, many pipelines transport hazard, flammable and toxic fluids, so even small incidents/failures in their operation can have really catastrophic consequences. But effective and safe operation starts from effective and safe design.
The engineering art of piping design is a difficult one, as there are lots of different technological, mechanical and safety requirements (some contradictory to each other) which piping designer has to comply with in order to propose optimal solution. And of course it is simply impossible to effectively design complex modern piping systems without powerful and easy to use software tools of analysis and simulation.

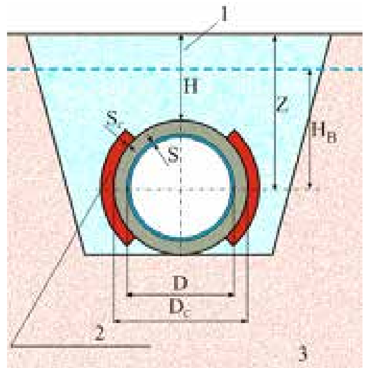


Fig.1 - Buried pipe cross-section. 1 - backfill soil, 2 - expansion cushion, 3 - foundation soil

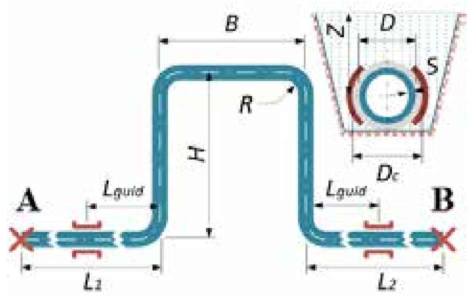


Fig.2 - U-shaped buried pipe expansion loop.

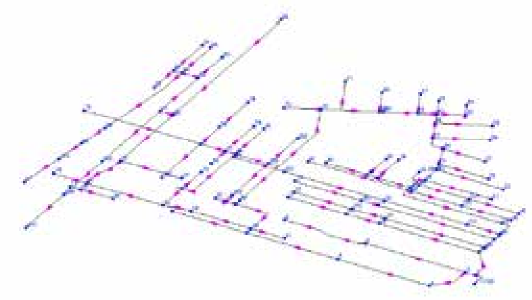


Fig.3 - Medium-size natural gas distribution network

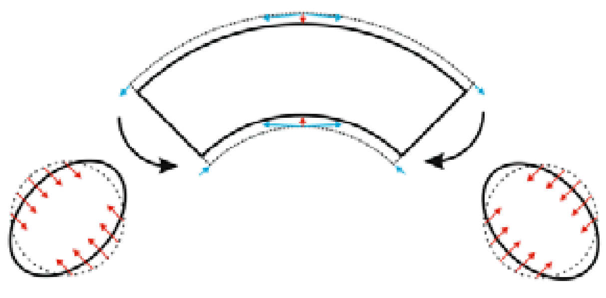


Fig.4 - Karman's effect

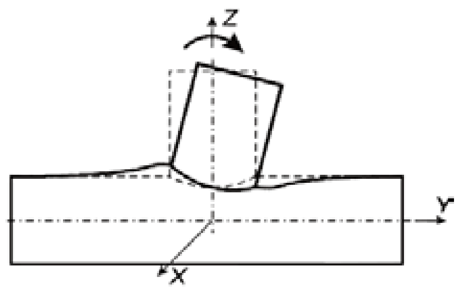


Fig.5 - Tee flexibility

So what is the problem – some readers (not familiar with piping design and analysis) can ask – currently there are so many powerful general-purpose FEM and CFD analysis software products, why not to use them to simulate such simple (as it can seem to one not involved into piping industry) objects as piping systems?
The problem is that piping systems only seem "simple", in fact they are quite complex objects with multi-level mechanical and mathematical description and with a lot of specific physical effects to be taken into account. So usage of general-purpose CAE tools for piping simulation would be expensive, time consuming and could give results far from reality!
To start with, piping systems as a rule should be modeled on at least 3 scale levels, with all these models linked together. And at each level it is necessary to select and use the "level of detailing" of mathematical modeling appropriate for the specific simulation task.
The 1st scale is local section of pipe (or specific piping "local" component like fittings, valves, piping supports etc). This is so-called "point model" (fig.1). On this scale level (which is of the order of one pipe diameter) it is required to describe and mathematically model both stress and strain pipe wall fields in pipe cross section and fluid velocity, temperature and stress fields inside the pipe. It is also needed to cover mechanical and thermal interaction with environment, including multiple layers of thermal insulation, electrical or steam tracing and interaction with air, soil (for buried pipelines) and water (for underwater pipelines). For multiphase (for example gas-liquid) flow it is also necessary to identify and describe flow patterns – how different phases are distributed across pipe cross section, and how they interact and move. On the same level of scale it is required of modelling hydraulic and thermal behavior of different piping components (minor pressure losses K-factors etc.), as well as of describing how different types of piping support work in terms of simple restrain conditions on pipe displacements. A description of pipeline ends (nozzle-vessel shell junctions) behavior is also needed. All this includes a number of interacting 1D and 2D (and sometimes even 3D for non-standard piping components) stress, fluid flow and thermal analysis tasks.

The 2nd scale covers objects from single pipe run through assembly of several pipes and piping components (for example U, Z and L – shaped loops – fig.2) up to piping system branch as a whole. The typical size for this scale is from ~10 to hundreds and thousands pipe diameters. On this level of scale it is necessary to describe how cross-section fields and average or integral cross-section values (pressure, temperature, fluid velocity, forces and moments in the pipe etc.) change along the pipe. This is 1D modeling task and can be described by a system of ordinary (for static analysis) or partial (for dynamic analysis) differential equations (with coefficient and boundary conditions taken from modeling on the 1st level of scale). On this level optimal parameters of some assemblies (for example piping expansion loops) should be calculated. Also this level of modeling includes flow assurance tasks – for example, prediction if some combination of pipe slopes would produce serious gas-liquid flow instabilities (severe slugging, terrain slugging), or if (and where) gas hydrate plug could appear and how to suppress these phenomena.

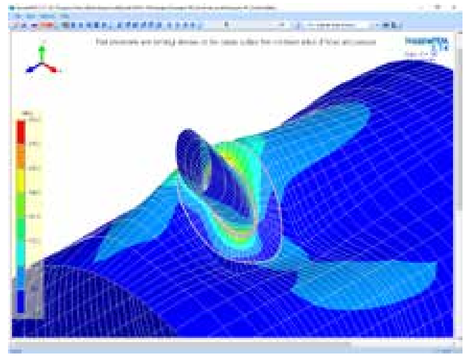


Fig.6 - Nozzle-FEM graphical postprocessor

The 3rd scale is the piping system as a whole (which can be mathematically described by a graph). Such large piping networks as municipal water or natural gas distribution systems, process and utilities piping systems of large process plants can be very complex, with hundreds and thousands branches, many loops and recycles (fig.3). The task of stress, fluid flow and thermal analysis of such piping systems on this level of scale can be formulated as a system of non-linear (as a rule) algebraic equations for variables in graph nodes using branch properties (taken from results of modeling on 2nd level of scale) and boundary conditions in source/sink nodes. And here is when modern graph theory and sparse matrix algorithms come into play, along with hydraulic network analysis methods. Also on this level sizing and parameter optimization problems for the whole network can be set and solved (what diameters to use, where

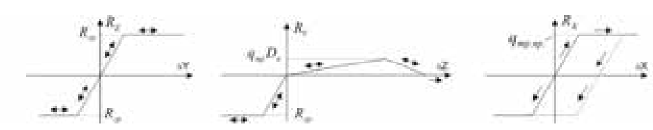


Fig.7 - Non-linear correlation of soil resistance and displacement in various directions

to place piping supports and which types of them to use. etc). And for simulation and prediction of existing piping systems behavior it is required to also solve model calibration problem – i.e. to restore real branch properties (for example pipe wall roughness) on the base on available real measurement data. This usually demands

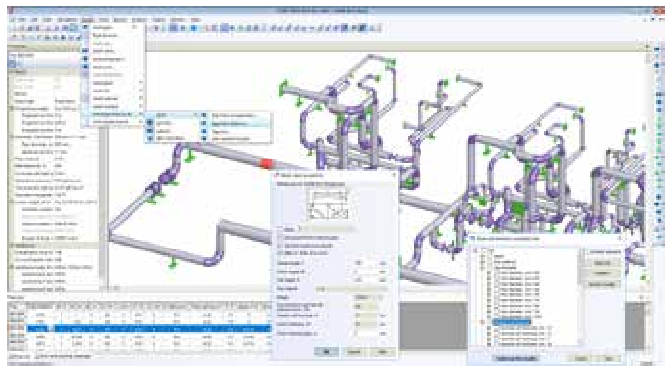


Fig.8 - START user interface

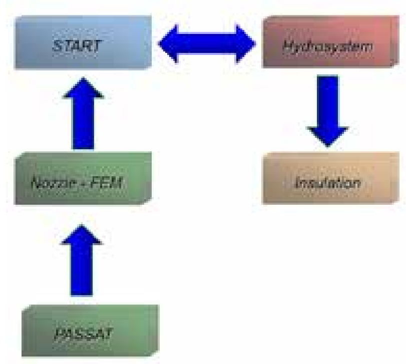


Fig.9 - Data transfer between PASS programs

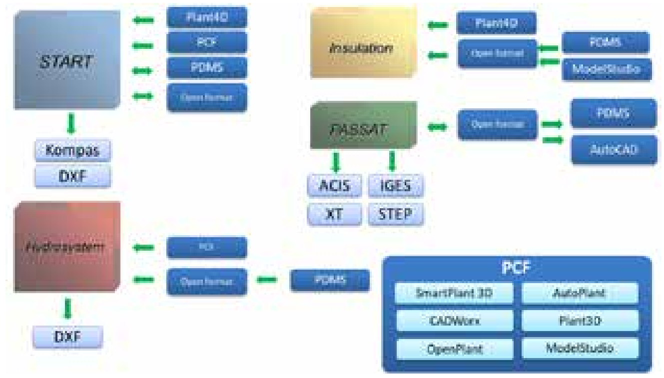


Fig.10 - Integration of PASS programs with CAD software

modern optimization methods (including genetic algorithms and others).
So now it is obvious how complex and multi-level piping system simulation is. But beside this, it requires taking into account many specific effects many engineers may be unaware of. In the following a short list of such effects by the example of piping system static stress analysis is reported.
At first glance all that would be necessary in this case is to model piping network as a set of connected pipe beams, apply restrains to pipe supports and end nodes (connections to equipment), use any appropriate stress solver and check the stresses. But the case is that much more aspects have to be taken into account.
First, it is necessary to consider internal or (for vacuum piping) external pressure. It completely changes stress field in pipe wall and can create thrust forces and displacement in pipelines with axial expansion joints. It also creates so-called bourdon effect that produces additional moments in bends with initial section ovalization!

Second, it is required of taking into account real flexibility of some piping components – otherwise results of simulation can be several times more conservative than in reality! This primarily concerns bends/elbows where so-called Karman's effect takes place – significantly increased flexibility due to cross section ovalization during bending (fig.4). Next it concerns flexibility of tees (fig.5) and (last but not least) flexibility of nozzle-shell junctions that also can significantly influence the results of stress analysis and in general case can be calculated only using appropriate FEM solvers (fig.6).
Third, piping supports. They often can have one-way restrains

Fig.11 - District piping network on background map in Hydrosystem

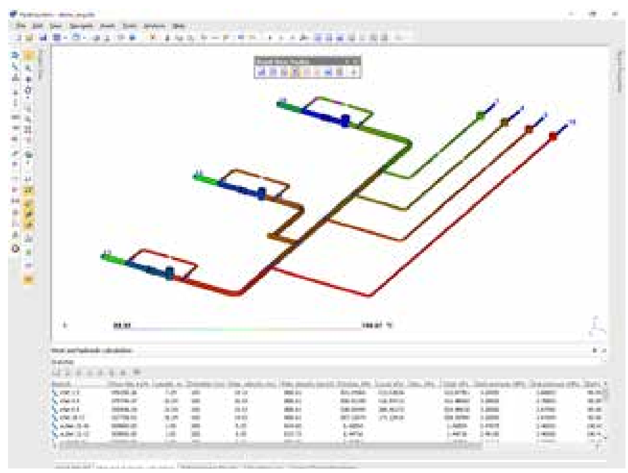


Fig.12 - Thermal analysis results in Hydrosystem (fluid temperature)

(sliding supports) or gaps, so can "switch on" and "switch off" depending of piping displacements. This immediately makes the problem non-linear. Moreover, friction forces on sliding supports can seriously influence piping behavior, and direction of these forces depends on changes of piping displacements in time – so the task becomes dependable on load sequence in time!
Fourth, for buried pipelines the interaction with soil should be included into the analysis. And the soil model is quite complex! It includes various non-linear soil resistance vs. pipe displacement laws, which are different in vertical, horizontal and longitudinal directions; change in soil resistance force for vertical and inclined pipes; change of soil resistance in flooded parts of pipeline and vertical buoyancy force in these parts; influence of pipe insulation and cushioning pads. (fig. 7)

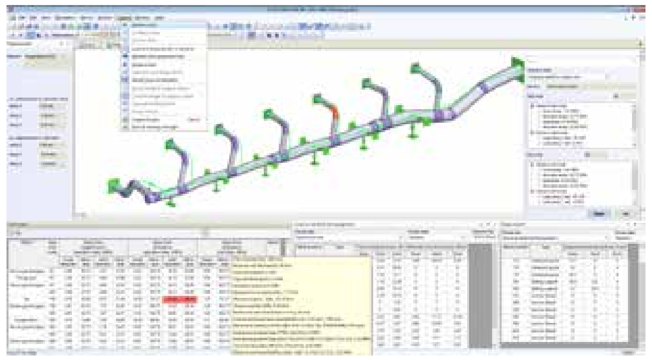


Fig.13

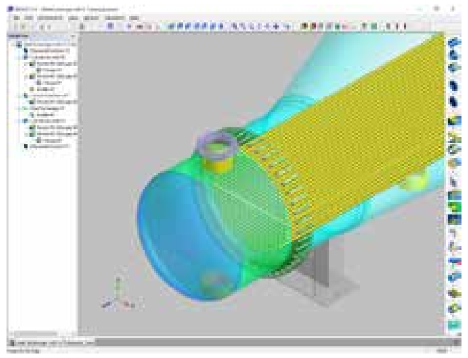


Fig.14 - Shell and tube heat exchanger in PASSAT

Fifth, for high-temperature fluid piping material behavior becomes non-elastic. Elastic deformations transitions into plastic (residual) deformations with creep and relaxation should be taken into account. And how to analyze results when (if) they are available? According to national standards (codes), which are different in different countries for different types of pipelines, it is mandatory to check stresses vs. allowable stress values for different pipeline states (corresponding to different combinations of loads) which can include typical sequence of states (installation state – operating state – cold state). In some cases it is also necessary to make fatigue strength analysis for typical temperature/operation cycle history. Plus it is necessary to check the value of displacements, value of forces and moments on piping supports and equipment nozzles vs. allowable values. And to check (when necessary) piping cross-sectional and longitudinal stability.
Hence, is it still possible to think that an average engineer can do such analysis in full scope in finite time using general-purpose FEM software?

So piping simulation requires highly specialized CAE software. It should incorporate all the knowledge in piping behavior and analysis power and deliver it to any process and piping engineer in user-friendly, easy to use form for every day work. To develop and support such software is this very mission which company Piping Research and Engineering Co (PSRE Co) follows for the last 25 years.
PSRE Co is, therefore, happy to introduce PASS – Piping Analysis and Sizing Software integrated suite – to all readers of EnginSoft Newsletter. It is already in active use by thousands of engineers in Russia, Ukraine, Kazakhstan, Belarus, China, South Korea, Japan, Czech Republic, Bulgaria, Lithuania, Germany, Italy, Spain, the Netherlands and other countries for analysis and design of process piping and plant utilities, district heat transfer, water and gas networks, oil and gas field gathering piping systems, and other types of pipelines. We hope that much more European engineers will now take advantage of its power with the help and support of our partners and friends from EnginSoft SpA.
PASS covers all main aspects of pipeline analysis – fluid flow and thermal analysis (Hydrosystem software) and stress, fatigue and stability analysis (START software). It also provides tools for vessels, columns, tanks and heat exchangers stress analysis (PASSAT software), stress and flexibility analysis of nozzle-shell junctions (Nozzle-FEM software), industrial thermal insulation calculation and design (Insulation software) and pressure relief system sizing (“Safety valve” software).
PASS integration features provide not only use of unified GUI (with synchronized pipeline/equipment model graphical window, model structure tree window, model component properties and calculation results panels and windows) (fig. 8) and shared software components, but also seamless data transfer between different PASS programs (fig. 9). Moreover, PASS can exchange geometrical data with the most popular general-purpose, mechanical and 3D plant CAD software (fig. 10)
Hydrosystem calculation model graphical representation can switch between different modes (PFD/P&ID – like unscaled scheme

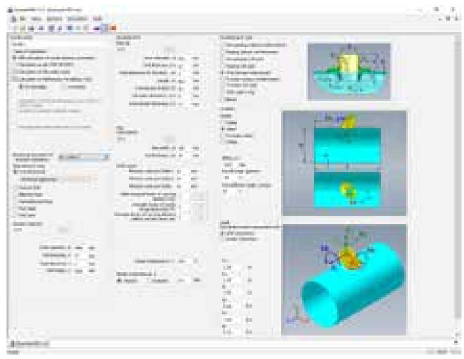


Fig.15 - Entering nozzle-shell junction data in Nozzle-FEM

with regulated level of detailing; 2D scaled distributed networks on background map; single line or solid 3D scaled or unscaled representation for process piping) depending on what is suitable for specific design phase and pipeline type (fig. 11). The results can be viewed as color maps (fig. 12), animation (for waterhammer) and easy-to-analyze tables and user customizable reports in PDF, MS Word, MS Excel and other formats.
START single line or solid 3D detailed graphical model can show results in stop-light (red, yellow, green) display and animate pipeline deformations (fig. 13), and stop-line colors of reports immediately attract user to problem results and piping elements.
PASSAT’s powerful C3D geometric kernel provides detailed 3D modeling of complex vessel elements (fig. 14) and integration via industry standard formats, and highly detailed result reports are ready for inspection submission.
Nozzle-FEM simple model enter panel (fig. 15) allows to define nozzle-shell geometry in a second, while power graphical postprocessor (fig. 6) and reports provide detailed review of results. PASS programs’ embedded help system and training movies instruct the customer not only how to “press the buttons”, but how to correctly build calculation model and interpret results, and even how to correctly use some piping components and constructions for optimal design (fig. 16). PSRE Co specialists also provide training in customer’s office.
PASS ease of use doesn’t mean lack of functionality. PASS programs have very powerful “engine” hidden beneath their beautiful user interface. And they take care of all specific piping effects themselves,

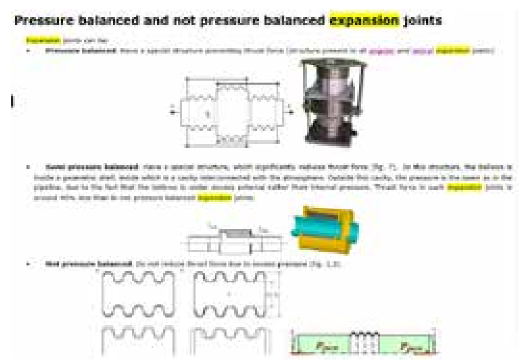


Fig.16a - START context help system explains different types of expansion joints…

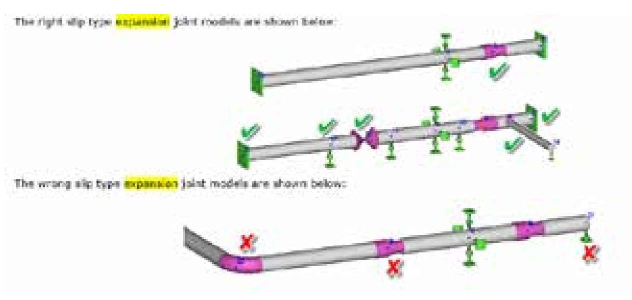


Fig16b -. … and how to install them properly in the pipeline.

so the user doesn’t need to be expert in piping systems!
Let us quickly look at some power features of 2 main PASS programs – START and Hydrosystem.
Hydrosystem provides heat loss and pressure drop steady state calculation for any complex piping networks for real liquid, gas and two phase gas-liquid flow. It also fulfils waterhammer calculations with cavitation (column separation) effect. Major and minor pressure losses are calculated based on Idelchik handbook of hydraulic resistance (the most comprehensive world-known research on the subject), also considering recent researches by Miller, Ito and other specialists. The flow rate distribution calculation uses most effective Global Gradient Algorithm (GGA) by Prof. Ezio Todini (University of Bologna, Italy), specially modified by PSRE staff for compressible and multiphase flow. Hydrosystem two-phase flow calculation is based on classical methods and modern “mechanistic” models, including the newest Tulsa University Fluid Flow Projects (TUFFP) Unified Model (TUFFP research group was formed and is supervised by famous multiphase flow researcher James Brill, and PSRE Co is a member of TUFFP). Automatic settings selection for TUFFP Unified model on the base of experimental database was specially developed by Hydrosystem’s authors in order to allow using this advanced model by non-experts. And as thermodynamics is the base of any process engineering analysis, PASS includes really powerful thermodynamics libraries: our own STARS library for fluid properties and phase equilibrium calculation by fluid composition with database of more than 1600 compounds and different petroleum fraction correlations; WaterSteamPro library for steam/water properties calculation according to the latest formulation of the International Association for the Properties of Water and Steam (IAPWS); the most advanced natural gas GERG-2008 thermodynamic library by Prof. Dr.-Ing. Wolfgang Wagner (Germany). PASS programs can also be used with one of the best world thermodynamic libraries – Simulis Thermodynamics from our partner ProSim SA (France). For gas hydrate deposition prediction special module from famous PVTSim library by Calsep A/S (Denmark) can be used.

START program provides stress analysis of aboveground and buried pipelines of any complexity under static, cyclic and seismic loads. It automatically takes care of all special effects mentioned earlier in the article (and many others!), as well as provides all necessary stresses, forces, moments and stability checks for states and load combinations to be analyzed according to specific selected national code. START utilizes the special, very quick and effective solver for this class of mechanical problems. For pipeline–soil interaction it uses very advanced Ainbinder’s model (instead of linear Vinkler one) with editable soil property database delivered along with the program. It can calculate pipelines with not only metal, but also plastic pipes and piping components. START also can easily analyze pipelines from such non-isotropic (orthotropic) materials as Fiberglass Reinforced Plastic (FRP), Glass Reinforced Plastics (GRP), Glass Reinforced Epoxy (GRE) according to ISO 14692-3 and contains database.

PASS programs provide users with not only pure analysis opportunities, but with different sizing and optimization features, making piping designer work much more effective and creative.
For example, Hydrosystem allows selecting optimal diameters of piping branches. It also can select optimal dynamic pumps with performance curve optimization via impeller trimming and speed adjustments (variable frequency drives); for this purpose Hydrosystem is integrated with famous European pumps selection software Spaix Pumps from our partner VSX – VOGEL SOFTWARE GmbH (Germany).
START automatically select optimal spring supports from embedded databases of such popular piping support manufactures as LISEGA, WITZENMANN and others. A special module START-Elements calculates pipe and fitting wall thickness; checks and sizes different typical piping assemblies (U, Z, L –shaped loops etc.) for both aboveground and buried pipelines; checks buried pipe longitudinal stability and easily solves many other everyday tasks of piping designer.
Of course it is impossible to describe in a short article all huge spectrum of PASS features which were being developed for decades. But with the help of colleagues from EnginSoft, PSRE Co hopes to inform Newsletter readers soon in much more details on PASS programs, their features and usage experience.
Please don’t miss PASS stories in next issues of EnginSoft Newsletter!

Leonid Korelstein, PSRE Co

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

TORQULESS
tremor absorber device

Introduction

A hand tremor is a symptom that is experienced through the shaking or trembling of the hands. This is usually caused by problems with areas of the brain that control movement. Various conditions may cause hand tremors, e.g. neurological problems, metabolic problems or side effects to different medications. Still, the main effect of hand tremors are the impact it has on a person's functionality, independence, life quality, and in many cases may cause shame and even depression.

Two undergraduate students from Ben-Gurion University of the Negev in Israel, have developed a unique device to absorb the tremor, thus dramatically improving the quality of life for those who suffer from hand tremors. Figure 1 shows a model of the TORQULESS design.

The basis of the design is a hand grip built from two parts. The first is fixed above the elbow and the second is fixed to the wrist. These two parts contain magnetic bracelets (Figure 2) which move relative to each other. Due to their special design, the relative motion of these two bracelets is damped by a magnetic field induced by the bracelets. Thus, the relative motion of the elbow and wrist is damped and the wrist shaking is dramatically alleviated.

Figure 1 - View of the TORQULESS apparatus

Each bracelet contains several small cylindrical magnets implemented in its perimeter. The magnet's poles are faced toward the other bracelet (as depicted in Figure 3) and by rotating one bracelet relative to the other, a resistance moment occurs due to the magnetic field imposed by each bracelet.

The magnetic strength is designed so that involuntary rotation of the forearm (as a result of vibrations) is limited, but allows normal rotation of the forearm when a critical torque (defined by the magnetic intensity) is overcome.

The design of the bracelets and magnets geometry, their strength and the way the magnets are scattered along the bracelet's perimeter was accomplished by numerical optimization using the software modeFRONTIER.

Figure 2 - View of the magnet "bracelet"

Design Challenge

Three challenges faced the design team

- Magnet strength limits
- Continuous movement
- Size

The magnets strength should prevent the unwanted tremor; however, twisting of the arm is possible if the applied torque exceeds the prescribed critical value. The one dominant factor affecting the magnet strength is the configuration of the magnets.

To maintain the capability of continuous movement, a maximum number of magnets must be spread evenly along the bracelet perimeter.

It is obvious that the handheld support device should be as small as possible to allow the device to be wearable under clothes.

To solve those three challenges, and to meet all requirements, optimization of various design variables is necessary and inevitable. These design variables are geometric, such as the number of magnets in each bracelet, the diameter and length of each cylindrical magnet, as depicted in Figure 4.

A simplification of the tremor was done by considering only hand rotation with the arm acting as a body which has a spherical joint. Pure torque represents the required tremor restrained was estimated as 0.3 N·m by an offline Solidworks simulation.

Two design goals were defined:

- The angle between each pair of magnets centers will be as small as possible, thus the number of magnets will be maximized. This ensures a smooth rotation of the TORQULESS
- The outer diameter of the bracelet will be as small as possible. This is done to minimize the dimension of the TORQULESS which will ease its conciliation.

Design implementation

modeFRONTIER was used to integrate and optimize 'Excel' and 'Matlab' scripts. Figure 6 depicts the design framework implementation within modeFRONTIER.

The Excel file included all the information about the various materials and mechanical properties of the layer which separates and reduces the friction between the two bracelets. Thus as part of the optimization the best material with the proper 'Friction coefficient', in compliance with constrains was found. The Excel file also includes all the information about 'Magnetic materials' from catalogs with the relevant properties. The Matlab script was used to calculate the various cost functions and constraints.

The selected optimization method was MOPSO (240 generations and DOE of 5 initial designs). Due to stringent constraints, only 255 feasible designs were obtained out of 1200. Other methods were tried but they obtained far fewer feasible designs.

Figure 6 shows the results output from the modeFRONTIER. The vertical axis is the first goal, namely the angle between two neighboring magnets. The horizontal axis is the second goal, i.e., outer bracelet radius. Each dot represents a different design and a distinctive Pareto front is shown as a compromise between the two goals.

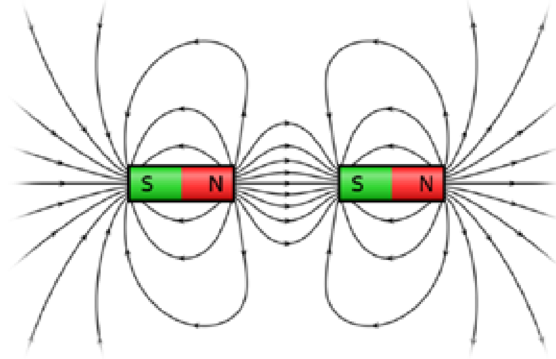


Figure 3, 2 - Magnets interaction

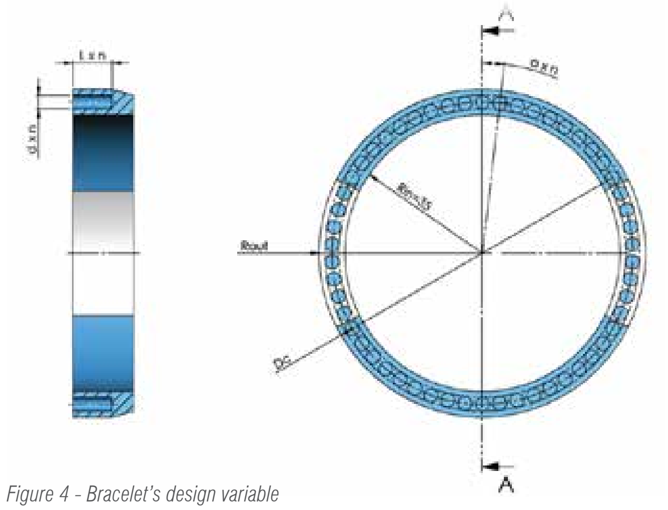


Figure 4 - Bracelet's design variable

The color scheme shows the design constraint of the level of torque above the critical torque (as a delta-torque). All design points are feasible and it is clear that the torque constraint is active for most design points along the Pareto frontier.

The chosen design is marked with red circle and represents the following design variable:

$n = 52$	*Number of magnets*
$d = 3.5[mm]$	*Diameter of the magnetic cylinder*
$L = 8.5[mm]$	*Cylinder Length*
$D_e = 78.1[mm]$	*Bracelet diameter*
$\mu = 0.2$	*friction coefficient between contact surface of the bracelets* *Polyethylene (polystone 7000SR)*
$B_r = 1.25[T]$	*Independent geometry magnetic field* *NdFeB 38/17*

Note that the final geometry is depicted in Figure 2.

Summary

Effective optimization is essential for multi-objective cases to find the best compromise between the objectives. modeFRONTIER was found to be very efficient in integrating different analyses from separate tools and establishing optimization design framework with little effort. The MOPSO algorithm was applied and proven as a very efficient method. This method is not "drawn" to a specific area but maintains diversity between solutions and scattered in the entire design space. In future development of the device, a high fidelity mathematical model will be developed using the software QuickField, which will then be incorporated into the process defined in modeFRONTIER. This will provide accurate values of magnetic force. The optimization was conducted by an undergraduate student with no experience of modeFRONTIER. Learning to use this powerful tool was quick and the students were able to experiment with various optimization methods with little effort. By that, modeFRONTIER is a perfect tool for optimization familiarization courses, both as a design tool and both as an "optimization educational tool".

Dr. Ohad Gur
Ben-Gurion University of the Negev

TORQULESS project was conducted by Mr. Boris Zborovsky and Mr. Shachak Netanel under supervision of Dr. Victor Weissberg.
The optimization implementation was part of the "Design Optimization Course" given by Dr. Ohad Gur

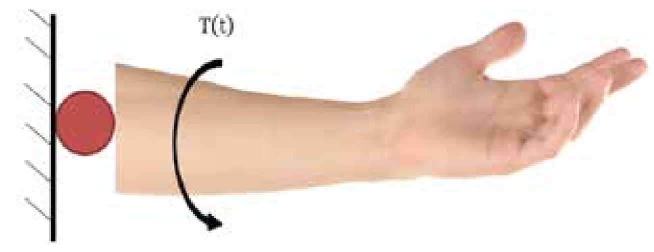


Figure 5 - Simplified tremor motion

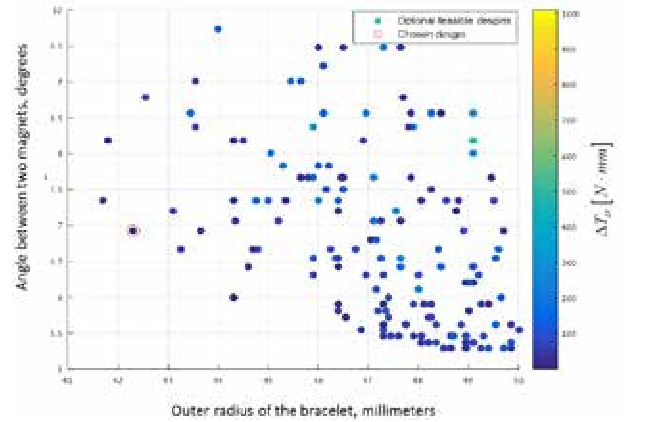


Figure 7 - Optimization results

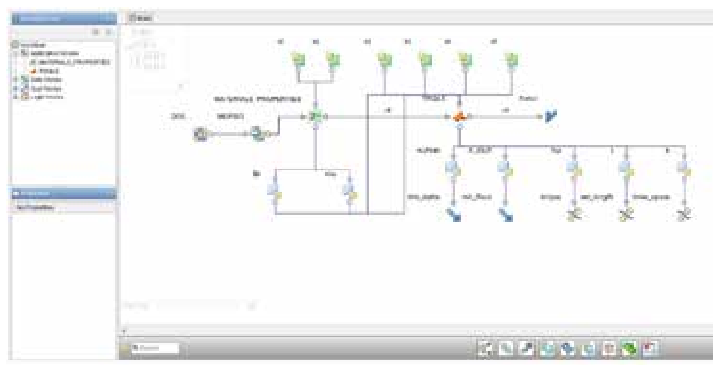


Figure 6 - modeFRONTIER implementation

Complex Parameter reduction optimization & RSM based tool for urban growth

Companies are continually looking for cost reduction by the implementation of smarter processes. This is often achieved through the purchase of new engineering tools which have to be assessed for return on investment. Often this assessment can be subjective and hard to quantify. What if current process integration and optimization techniques could create new, bespoke, game changing tools for a company?
While the benefits of standard optimization techniques are being realized the world-over, there are new benefits to be gained by using the latest generation of highly scalable process integration tools and optimization algorithms.
EnginSoft UK have collaborated with Anglian Water and Atkins to develop a new methodology for creating a predictive RSM based tool that can reduce housing development cost enquires from weeks to a day. This paper as presented at the NAFEMS European Optimization Conference 2016 demonstrates the principle steps taken to develop the tool and highlights the key technical decisions made to ensure its predictive quality.
This article (an excerpt from a paper abstract) briefly shows how ad-hoc design optimization can be replaced by an up-front full exploration of the design space using a computationally intense DoE of optimizations. It shows how the problem can be reduced using modeFRONTIER to perform a complex parameter reduction optimization.

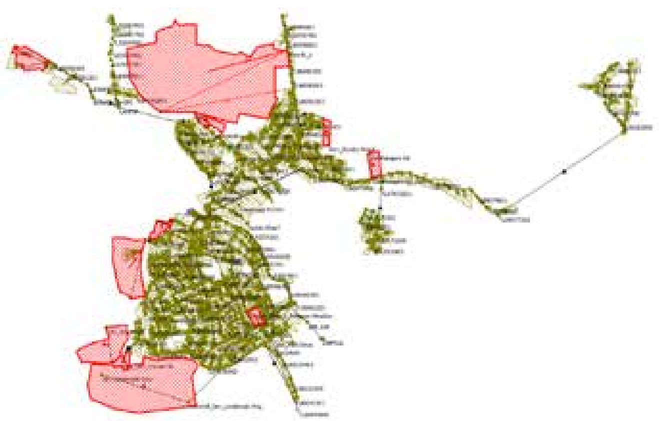


Fig. 1 - Housing Development Locations in a Catchment

Problem

Each area of the UK has a local housing plan with areas of land ear-marked for development. The development of these sites are done at the discretion of the developers and local planners. Any detriment to the drainage system needs to be mitigated by upgrading the sewer network. The order in which they are developed is unknown and can cause issues in strategic planning for the catchment. See figure 1.

Often a series of lower cost localized, upstream solutions are found for each development when a more strategic higher cost downstream solution could be better in the long term.
The problem is that in a city there can be over a 100 development sites, and for a truly strategic approach, a solution is required for every combination of development sites possible. Computationally, it is impossible to perform this analysis manually, especially because every development scenario requires an optimized solution.

The Approach

Therefore an approach has been developed that reduces the number of design variables to minimize the time to find an optimized solution and utilises accurate RSMs to predict the solution cost for any development scenario.
The bold approach is to run a DOE of optimization, on top of which RSMs are trained, see figure 2.

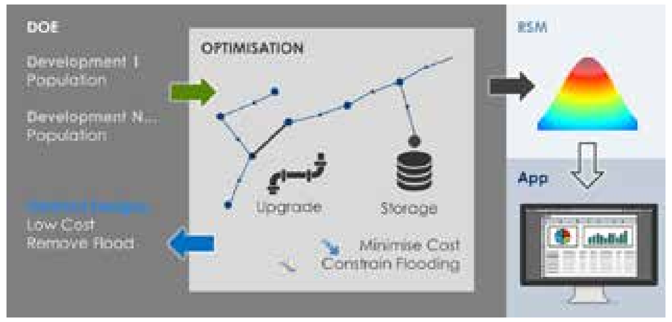


Figure 2 - Optimization Approach

Figure 3 - Urban Growth Prediction Tool

The Tool

The final result is a desktop tool/app that can be used to assess the development enquires as they are made. The user simply sets the particular development scenario proposed and by interrogating, the underlying RSMs can predict a costed, feasible solution in seconds, see figure 3.

This has the potential to allow an instant response on whether or not the proposed development will cause detriment to the sewer network and what the cost will be to solve.
Traditionally, a water modeler is required to assess the impact of a development each time an enquiry is made. This maybe done many times over the course of a year and involve a delay to the developer in obtaining a response.
With this type of new tool the urban developer can have direct and immediate answers about the risk, solution and cost of developing a particular site, see figure 4.

Figure 4 - Optimization Approach

By performing a comprehensive analysis upfront, savings can therefore be made in the long term. This has all been achieved through the unique power of modeFRONTIER to handle complex processes and large datasets.

John Barnes
Technical Manager EnginSoft
j.barnes@enginsoft.com

EnginSoft & SIMAI

SOCIETÀ ITALIANA DI MATEMATICA APPLICATA E INDUSTRIALE

The role of SIMAI (Società Italiana di Matematica Applicata e Industriale, http://www.simai.eu/en/) is to promote mathematic research in industrial and applied science, offering a view of applied mathematics that can be transferred from academia to industry.
The partnership between EnginSoft and SIMAI has been confirmed for the next 4 years; being SIMAI partners has important implications, in that it allows to promote research in the applied and industrial mathematics field, by linking the academic world with industry.
New industrial partnerships are always welcome to SIMAI, because they help bringing new ideas, networking and actual industrial needs to the fore. Manolo Venturin, representative for EnginSoft in the SIMAI Board, and his group in Padua are also involved in such activities.

Inverse analysis approach for determining the superplastic behaviour of the Titanium alloy Ti6Al4V-ELI

Introduction

Superplastic Forming (SPF) is considered one of the most suitable innovative solutions when highly complex components need to be manufactured; but a Finite Element (FE) approach is always necessary if components characterized by optimal thickness distributions are requested. Among the key aspects to be correctly defined for the creation of a reliable and robust numerical model, the description of the material behaviour certainly plays a key role. For this reason, the characterization methodology should induce a strain condition similar to the one experienced by the material during the SPF process (free inflation test, conical die test or multi-dome test are examples adopted in literature). In addition, such methodology should be accurate and fast at the same time in order to: (i) limit the quantity of the material needed for the characterization; (ii) be easily adopted to characterize each new batch (since the chemical composition or the mean grain size not always give enough information about the material properties).

In the last years, the evaluation of material parameters using experimental tests has been demanded to an inverse analysis approach, often driven by optimization algorithms: the simplex method or the gradient-based algorithms are reported to be viable solutions even though being sometimes stuck in local minima (maxima). On the contrary, Genetic algorithms (GA), since starting from an initial population and creating successive generations by means of selective operators, are capable to overcome the above mentioned limitations.

In the present work, the characterization of the superplastic behaviour of the Titanium (Ti) alloy Ti6Al4V-ELI (Extra Low Interstitial), largely adopted for biomedical applications, was carried out by means of free-inflation tests: both tests at a constant pressure and alternatively varying the gas pressure between two levels (JP test) were performed. Material constants of the Backofen equation, widely adopted to effectively describe the superplastic behaviour, were evaluated by means of an inverse analysis approach driven by a genetic algorithm: the dome height evolution during the tests was acquired and compared to the one coming from a simple 2D FE model using the commercial software ABAQUS, with the aim of minimizing the error between the two. Since the FE model was light enough, thousands of simulations were automatically run (integrating the Abaqus solver within the optimization procedure): reliable material constants after approximately 20 hours of computations could be thus obtained.

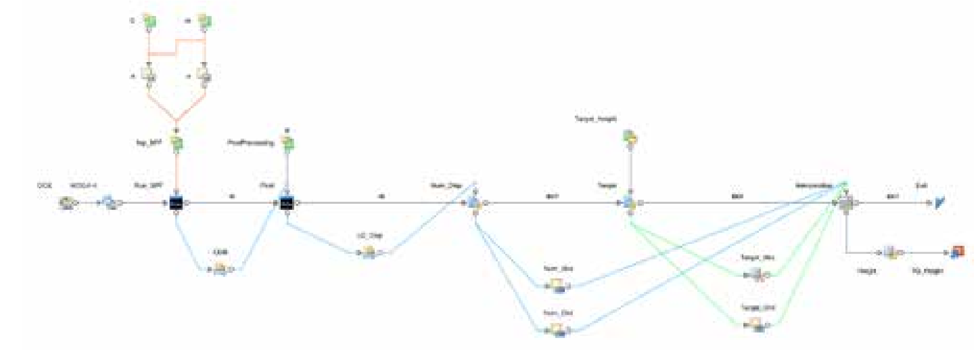


Figure 1 - Schematic overview of the modeFRONTIER workflow

Experimental tests & Inverse Analysis

Free inflation tests were performed at 850°C on circular specimens (D=80mm) extracted from a 1 mm thick Ti6Al4V-ELI sheet. The laboratory scale equipment was specifically designed to be mounted on an INSTRON electromechanical testing machine; the Ti specimen was clamped between a cylindrical die and a blankholder and deformed using pressurized argon gas.

Free inflation tests were carried out using two methodologies: (i) setting a constant pressure (indicated as CP tests); (ii) changing the pressure alternatively between two different pressure levels (indicated as JP tests). CP tests were conducted at 0.5 and 1.0 MPa, while pressure was varied between the same levels during the JP test. The dome height evolution according to time was continuously recorded during the tests.

The experimental setup was modelled within the Abaqus/CAE environment by means of an axisymmetric model; the material behaviour was defined using the Bailey-Norton power law $\varepsilon = A\sigma^n$ available in ABAQUS (material constants (A,n) were directly related to the constants (C,m) of the Backofen constitutive equation $\sigma = C\varepsilon^m$).

The evaluation of the material constants was demanded to an inverse analysis approach aimed at minimizing the difference between the experimental dome height curve acquired during the JP tests and the correspondent numerical one calculated using the above described FE model.

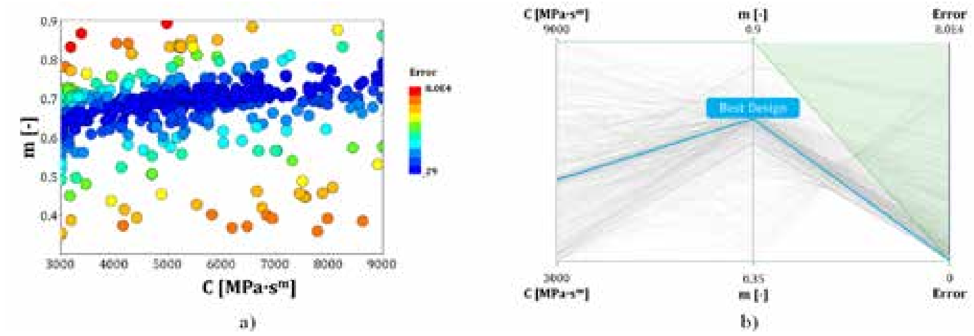


Figure 2 - Results of the inverse analysis: a) Bubble Chart and b) Parallel Coordinate Chart

Figure 2 depicts the workflow in the modeFRONTIER environment (v 4.5.3). The material constants of the Backofen equation were set as the two input parameters of the inverse analysis: at each run, once defined C and m, the correspondent values of A and n were determined by means of two transfer variable nodes and copied within the Abaqus input file.

Each simulation was run in batch mode (Run_SPF DOS node) and the results file passed by means of a transfer file node (ODB) to the second DOS batch node (Post) to carry out the post-processing according to the commands provided by a python script file (PostProcessing input file node). The extracted numerical dome height curve was subsequently transferred in the Calculator node (Interpolation) to be compared with the target curve, i.e. the one recorded during the JP test. At the end of each run, the error between the two curves was calculated. The inverse analysis, aimed at minimizing the above mentioned error (by means of the target node TG_Height), was carried out adopting a multi-objective genetic algorithm (MOGA-II) starting from an initial population of 50 individuals, created with the Sobol algorithm (in order to avoid design clusters), and considering 20 successive generations. Simulations were run on Xeon 3.47 GHz dual processor with 40 GB RAM installed, taking less than one day (20 hours and 26 minutes) to complete the whole optimization (1000 designs).

Results and discussion

Results obtained from the inverse analysis have been summarized in Figure 2. The bubble chart (Figure 2a) shows that the designs characterized by lowest values of the previously-defined objective function (blue bubbles) are approximately located in a range of m between 0.6 and 0.75 and in a range of C between 4000 and 6000 MPa•sm. In addition, the parallel coordinate charts in Figure 2b, since created narrowing the Error axis up to the minimum value, allows to highlight the best design, i.e. the one characterized by the best fitting between the numerical and experimental dome height curve.

The obtained values of C and m were compared to those coming from the analytical model proposed by Enikeev and Kruglov (using the dome height curves coming from the two CP tests at 0.5 and 1.0 MPa).

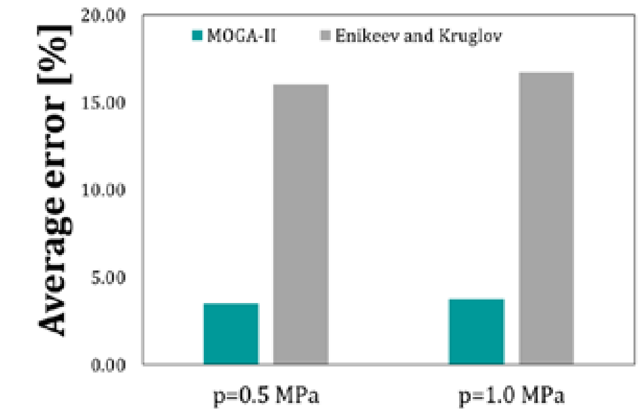


Figure 3 - Average errors relative to the validation runs

To finally validate the proposed approach, two load conditions (concerning the CP tests conducted at the pressure of 0.5 and 1.0 MPa) were simulated implementing either the material constants coming from the inverse analysis and the ones coming from the analytical model proposed by Enikeev and Kruglov. Numerical dome height evolutions according to time (in both the approaches) were compared to the ones acquired during the experimental CP tests: the material constants coming from the inverse analysis revealed to be able to better reproduce the material behaviour in both the simulated load conditions, as shown by the average errors (in percentage) plotted in Figure 3.

Conclusions

Material constants of the Backofen power law have been evaluated in the present work by means of an inverse analysis approach driven by a multi-objective genetic algorithm: a simple 2D axisymmetric numerical model was used for simulating the experimental setup of free inflation tests conducted changing alternatively the gas pressure between two levels. The genetic algorithm gave back the optimal value of the material constants able to minimize the error between experimental and numerical data (i.e. the evolution of the dome height according to time) acquired during the tests. The final validation adopting two load conditions (different from the ones used for the calibration of the material model) confirmed that: (i) the adopted experimental methodology resulted to be highly suitable for the inverse analysis; (ii) the material model set using the inverse analysis approach allowed to optimally fit experimental data.

Material Constant	MOGA-II	Enikeev and Kruglov
C	5229 MPa s^m	6421 MPa s^m
m	0.703	0.746

Table 1 - Material constants obtained from the inverse analysis and from the analytical approach

Acknowledgements

The activities in this work were funded by the Italian Ministry of Education, Universities and Research Government through the PRIN Project 2012 "Biomedical Titanium alloy prostheses manufacturing by means of Superplastic and Incremental Forming processes" (project acronym: BIOFORMING).

A. Piccininni, P. Guglielmi, G. Palumbo, L. Tricarico
Politecnico di Bari

D. Sorgente
Università degli Studi della Basilicata

Advanced Joining Techniques simulation and optimization using LS-DYNA and modeFRONTIER

Increasingly stringent government fuel consumption legislation and evolving consumer preferences are making a difference. Automakers are achieving weight reduction by replacing lighter materials such as aluminium, magnesium and advanced composites as well as making vehicle design changes.

Therefore a maximum flexibility of the used materials is necessary and new joining techniques are constantly developed.
Self-pierce riveting technology is a relatively new fastening technique. It has become a potential joining approach for aluminium alloy structures in the automotive and aerospace industry owing to their advantages over conventional fastening methods, such as spot welding.

The advantage of a self piercing riveted joint in contrast to other joining methods such as spot-welding is to combine materials with different properties such as aluminium and steel.
Also, the joining process of self-piercing rivets does not affect the microstructure and thus the properties of joined materials.

Failure of a joint depends on various factors such as the geometry of joint configuration, sheet strength that are joined, rivet material used, cracks developed during joining, and many other.

This paper summarize a work divided into three stages. In the first stage, a 2D simulation of riveting process has been carried out over two sheets made of steel and aluminium alloy. In the second stage of work, it has been conducted a numerical-experimental correlation in order to validate the simulation results. In the third stage, the rivet and die virtual models have been parameterized in order to perform a shape optimization.

Numerical simulation has been used to investigate the riveting process. LS-DYNA is a powerful tool for performing an adaptive meshing simulation in order to reach a higher accuracy of results avoiding high elements distortion. The main goal of the simulation has been to locate the critical areas in which crack and fracture can occur in the original configuration. The study has been carried out to guarantee the integrity of the rivet and the joined sheets.
The target of the project has been to develop an automatic and integrated approach to study the best rivet-die geometrical shapes.

Introduction
Modern vehicles are made of different material types: steels, aluminium alloys, magnesium alloys, polymers. As conventional spot welding cannot be used to join part made by different material, a new kind of technology suitable for this purpose is the self-piercing riveting. This is mainly used in automobile and aircraft industries, and consist in a single-step process of joining without any predrilled holes. A semi-tubular rivet made of high strength steel is used to make a joint between sheets.
The simulations were performed using an explicit-implicit solution technique with mesh adaptivity.

LS-DYNA is a powerful tool to simulate the SPR process due to several reasons:
- adaptive meshing features;
- several material models implemented;
- robust contact algorithms;
- springback analysis (explicit-implicit coupling).

Numerical simulation with LS-DYNA virtually assess the joining quality and permits to evaluate stress, force and deformation during joining.

After virtually testing the original configuration provided by the manufacturer, the rivet-die geometrical shapes were automatically modified in order to find a better solution.

The current paper will show the main aspects regarding the development of an automatic approach to execute these tasks.

SPR model description
TThe SPR joining technique relies on creating a mechanical interlock by flaring the hardened steel rivet shank, after it pierces a second sheet in joining a two-sheet assembly. The technique is not restricted to joining a two-sheet assembly, as it may be used to join three or more sheets comprising different materials in a single operation.
Self-piercing riveting is a large-deformation process that involves piercing. The self-piercing riveting process can be summarized by four steps:
1. clamping: the blankholder presses the two sheets against the die, and then the rivet is gradually pressed into top sheets
2. piercing: the punch pushes the rivet into the top sheets
3. flaring: the material of the lower sheet flows into the die and the rivet shank begins to flare outward, forming a mechanical interlock between the upper and lower substrates
4. punch release: once the punch is retracted, the finished joint is achieved with the fastener properly seated in the sheets.

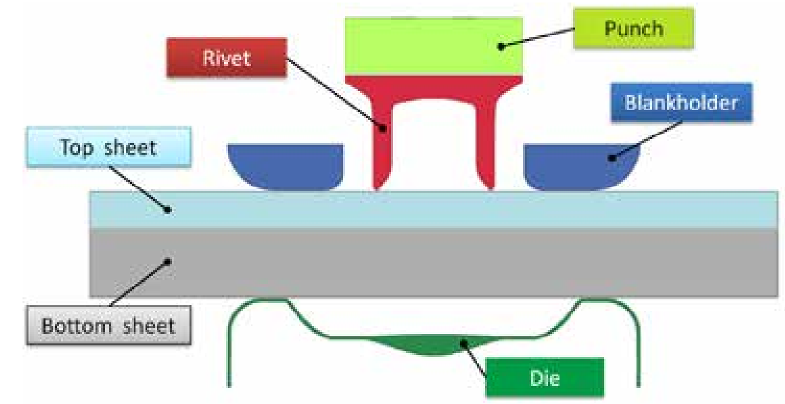


Fig.1 - SPR tool setup

The process comprises many pairs of contacts between the punch, blankholder, rivet, top sheet, bottom sheet, and die.
The rivet material is a boron steel, top sheet is a dual phase steel, while the bottom sheet is an aluminium alloy.
Blankholder, punch and die were treated as rigid bodies.
Regarding contact interfaces, six pairs of contacts have been created to handle the friction distinctly. For the 2D simulations conducted *CONTACT_2D_AUTOMATIC_NODE_TO_SURFACE contact has been chosen.

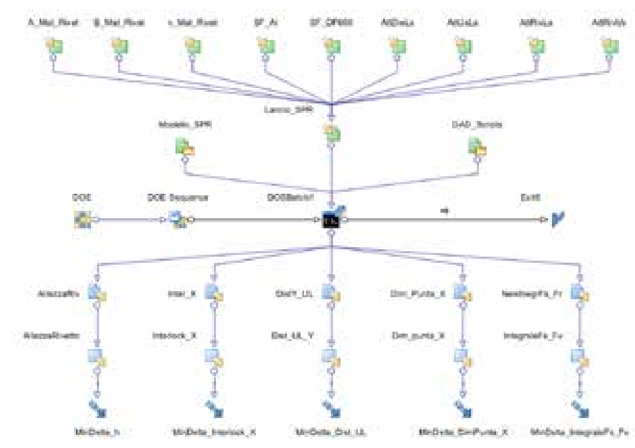


Fig.2 - Example of modeFRONTIER workflow used to perform the model calibration

Frictions and materials calibration
Friction coefficient and material properties were not easy to set for simulation for the following two main reasons:

1. the materials data received by the customer concern tensile test and do not taking account treatments and machinery;
2. the friction typology change during the process.

Preliminary simulations conducted have shown that friction play an important role in the penetration of rivet into sheets and has an influence on the results of the simulation. In particular, final shape of the rivet shank and the part of the top sheet in contact with the tip of the rivet shank is influenced by friction.

In order to find the correct parameters to use in the simulation, frictions and materials have been parametrized and a modeFRONTIER workflow built.
Experimental laboratory test results have been carried out to define all the materials and friction coefficients using the modeFRONTIER Integration and Design Optimization software.

For rivet material is being used Material Type 98 implementing Johnson/Cook strain sensitive plasticity, in which the flow stress is expressed as follow:

$$\sigma_y = \left(A + B\bar{\varepsilon}^{p^n}\right)\left(1 + C\ln\dot{\varepsilon}^*\right)$$

It is typically used for problems where the strain rates vary over a large range. The values of the input constant A, B and n define, respectively, the yield stress and hardening parameter and they have been set as input variables to find the correct material behaviour, considering treatment and machinery.
The same procedure has been implemented for the aluminium (bottom sheet) and steel (upper sheet) but using elasto-plastic materials Material Type 24 and parameterizing the stress versus plastic strain curves.

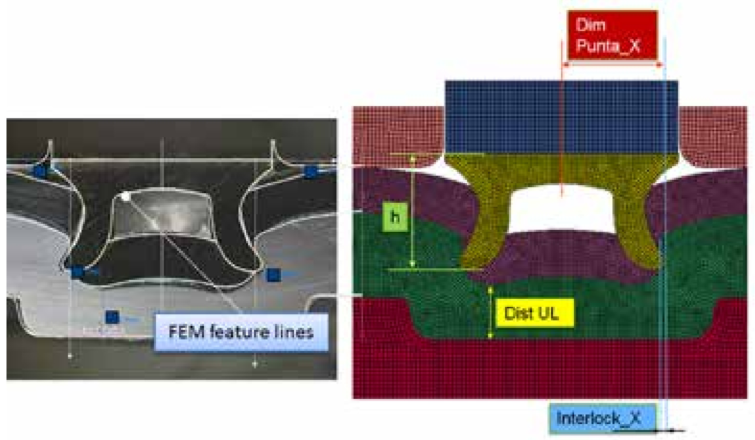


Fig.3 - Overlapped image - Experimental vs. Numerical

Adaptive meshing

During riveting simulation, high elements deformation occurs and can cause “error termination”. To avoid this, a 2D remeshing technique has been used.
In particular, this capability is called “rezoning” and consists in three steps:

1. generate nodal value for all variables to be remapped;
2. rezone one or more materials;
3. Initialize remeshed regions by interpolating from nodal point values of old mesh.

Important card to set this feature in LS-Dyna environment are *CONTROL_ADAPTIVE, *CONTROL_ADAPSTEP, *PART_ADAPTIVE_FAILURE.

Post processing

As explained in the previous paragraph, the mesh change during simulation, thus the nodes ids change continuously and so It is difficult to measure a distance between two nodes in automatic way (using a macro).
For this reasons, an ad hoc software has been written; it recognizes the geometrical characteristics at the end of riveting simulation and make a comparison with the section obtained cutting the real junction.
The geometrical measures chosen to make the comparison are highlighted in Fig 3; during the post processing phase, the software have to compare: interlock, rivet deformed height, distance between upper sheet and lower sheet and maximum rivet diameter calculated on the rivet tails. In addition the post processing compares riveting force.

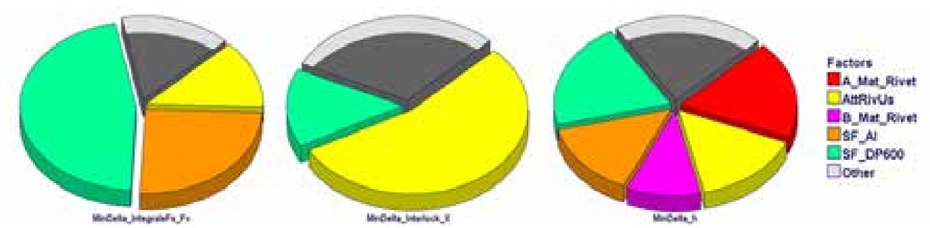


Fig.4 - Overall Student Chart – Output vs. Input correlation

DOE sensitivity analysis

Parametrical study were conducted to investigate how friction value among parts and material characteristics affect the joining results.

By means of this system, a DOE sensitivity analysis has been carried out in order to:

• Investigate material changes caused by treatments and machinery;
• Calibrate friction coefficient;
• Find correlation among objective functions verifying possible redundancy, for select independent function to use in the optimization phase;
• Find correlation among input variables and objective functions, to investigate how the system reacts to an input variable perturbation and define the ones that have major influence on objective function.

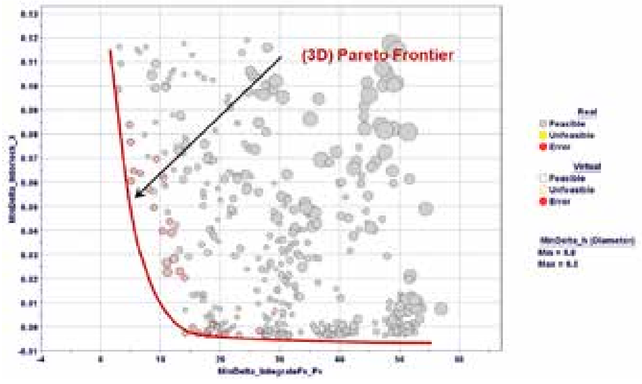


Fig.5 - Trade-Off Objective Functions

The Overall Student Chart highlights, by means of pie charts, the influence of each input (slice) on each output (pie).
In this case it emerges that the friction between rivet and upper sheet is very influencing for all objectives, in particular for the interlock.
To match the experimental test force, instead, the most important parameter is the top sheet material scale factor on stress-strain curve.
Rivet deformed shape is affected by all input variables investigated, practically in the same measure.
To choose the best combinations of the input variables that match the experimental tests involve an approach that considers a “trade off” concept, considering three objective functions.
In fig.5, it is possible to see the data in three dimensions. It cans be clearly noticed that the best designs are those close to the Pareto curve (best compromise interlock vs. riveting force), with small diameter (rivet deformed height). The parameters calculated for this specific case have to be a general validity. If not, the FEM model is correlated for the single set up but is not representative of the SPR joining process in general.

In order to validate the parameter to use in the SPR simulation, different analyses have been carry out.
The following scenery have been calculated:
• Upper and lower sheet with different thicknesses;
• Joining a three-sheet assemblies;
• Different rivet geometry.
The simulation results have shown a good agreement with laboratory tests and so the parameters obtained have been used to perform the optimization phase.

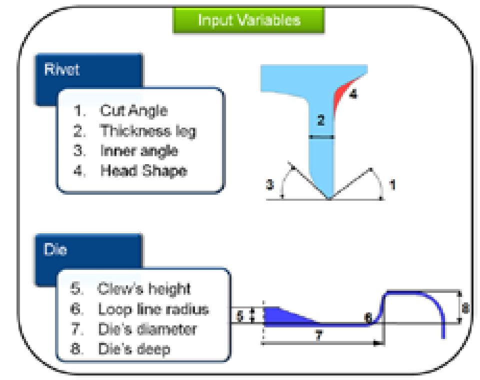


Fig.6 - Morphing input variables

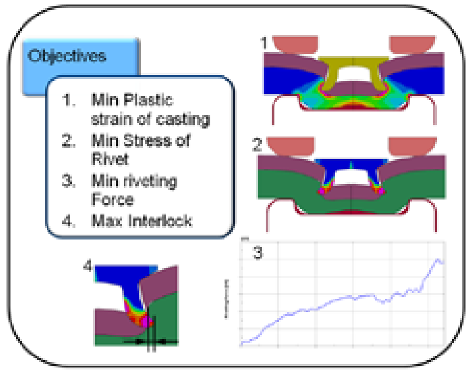


Fig.7 - Morphing objective functions

Optimization strategy definition

The previous tasks provide a simulation that represent, with a good agreement, the real riveting process.
It is possible now to modify the shape of the rivet and die and investigate the effects on the SPR junction.
In particular, it has been decided to change the angles of the tail, the thickness and the head shape for the rivet, while for the die the morphing parameters are: clew’s height, loop line radius, diameter and deep. The rivets producer confirmed the feasibility of all the shapes that this parameters combination could generate.
The goal of the optimization phase is to find designs that satisfied the following objectives:

- Interlock maximization, to improve joining resistance
- Riveting force minimization to reduce risk of joining failure
- Reduced plastic deformation on the bottom sheet to avoid bottom cracks
- Reduced rivet stress to avoid fracture on the rivet tails

In order to perform the optimization, modeFRONTIER has been used. modeFRONTIER is a state of the art multi-objective and mulltidisciplinary optimization software.

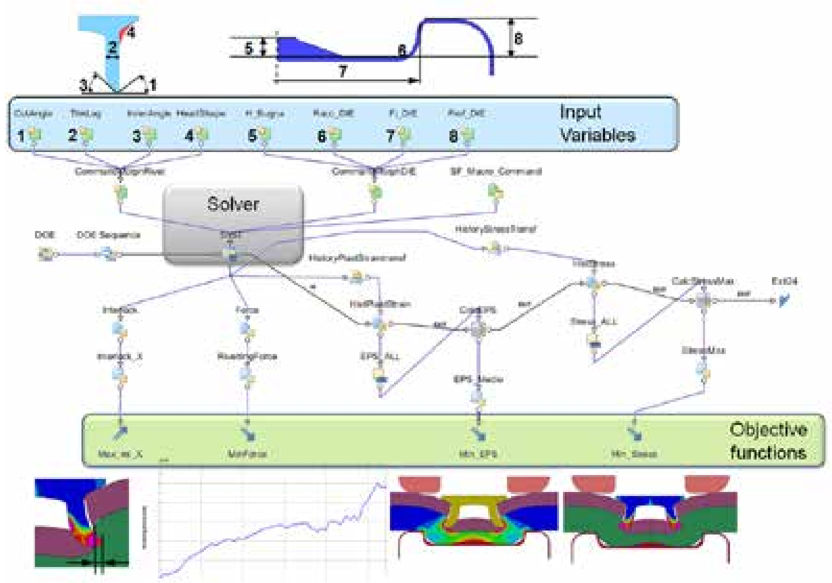


Fig.8 - Logic flow implementation

In the presented work, the input variables are the morphing operations, so a change in the value implies a modification of the rivet and die shapes.
By means of the “sh node” it is possible to invoke both LS-DYNA solver that submits the analysis and LS-PrePost that reads the output files and generates further result files suitable for mF interpretation.
Such results are values that mF handles and, thanks to internal optimization algorithms, modifies as modeFRONTIER input variables to achieve the target outputs. Extracted values can be also managed inside a calculator node where further operations can be performed.
By interpreting the workflow in fig 8, it is possible to identify all the methodological processes. Starting from the top, it is possible to see the input variables related to the morphing phase; a transfer file is created and passed to the LS-DYNA node. The support file folder, containing all files needed for the simulation and for the post processing also converges at this node. Moving forward in the flowchart, the output template node needed to read the simulation results is represented. Outputs, the objective functions that have to be to minimized or maximized to achieve the target results.
Due to a large amount of information related to the optimization tasks and the complexity of the methodology/work flow, the optimization results and the discussion will be the subject of a further article.

Conclusion

The final goal of the activity is to design the rivet, die and the riveting process. Actually, the car makers choose the rivets from the supplier catalogue and the approach is “passive”; clearly the supplier pushes the already existing rivet geometries in order to avoid new dies machining and production process set up. But, probably, the catalogue doesn’t include the optimum rivet for the specific application.
The developed methodology permits to identify, between the feasible geometries, the best shapes for the rivet and the die from the joining process point of view avoiding failures and maximizing the junction strength.

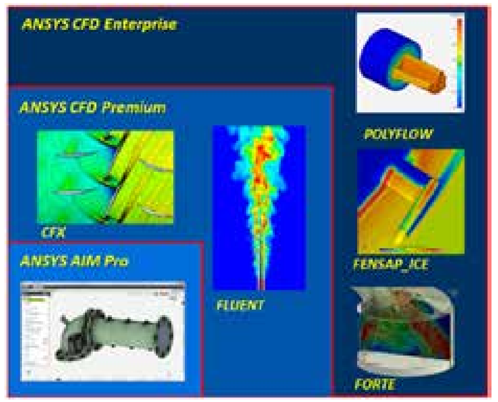
C. Martin, A. Ortalda
EnginSoft

For more information: Alfonso Ortalda, EnginSoft
a.ortalda@enginsoft.com

ANSYS CFD R 18

ANSYS 18.0 fluid dynamics makes now possible to take advantage from CFD simulations for all engineers with different level of experience thanks to the new fluids packaging, that provide the best capabilities in simple packages tailored to each user. The ANSYS Fluids product line is now composed by three levels:

- ANSYS AIM Pro: easy simulation environment designed for all engineers, that offers integrated CFD and other single and multiphysics solutions
- ANSYS CFD Premium: bundle that includes ANSYS CFX and ANSYS Fluent for experienced engineers that need well-validated CFD models across a wide range of general and multiphysics application
- ANSYS CFD Enterprise: comprehensive tool kit with all the ANSYS CFD Premium capabilities plus AIM-Pro and specialty solvers and analysis tools (Fensap ICE, Forte, Polyflow) for experts in CFD and large organizations.



For Premium and Enterprise Solvers the engineers can use 4 cores without requiring additional HPC licenses and first HPC Pack provides 10 cores.

ANSYS AIM R18.0

ANSYS AIM is a complete simulation tool that combines intuitive and guided workflow with accurate results and optimization features. It includes electromagnetics, thermal, structures or fluids or a combination thereof. Design engineers can perform upfront simulation as part of the product development process and send their models to simulation analysts to perform more advanced simulations, including additional physics. New features include magnetic frequency response, solution dependent expressions, one-way FSI for shell structures,

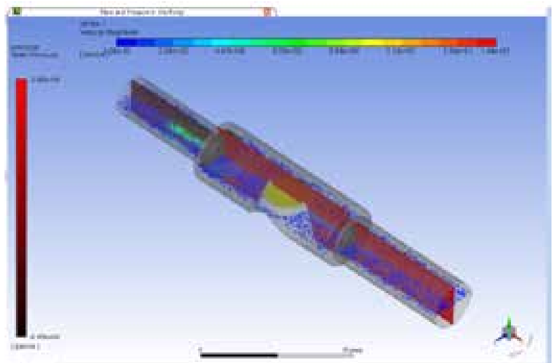
spatially varying force and pressure, multiple results display. With the new realease ANSYS 18, AIM can transfer mesh and selection set data from AIM to Fluent (or Mechanical) via Workbench project schematic connection.

ANSYS FLUENT R18.0

ANSYS FLUENT is the flagship product for general purpose CFD simulations and with the last release offers new capabilities and enhancements for a more comprehensive approach for the product design.

Fluent User Interface

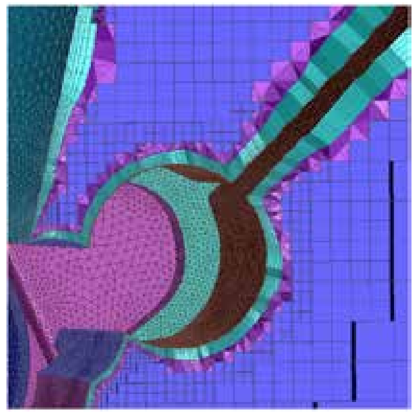
The graphical interface consists of additional features in order to improve the workflow, in particular for large complex cases, with new wildcard filtering of lists, drop-downs, grouping and multiple selection. The monitors are replaced with more flexible report definitions with better performance and the possibility to include discrete phase reports, algebraic expressions, custom field functions; a new cell register can be used for the identification of the problematic



cells during solution iteration. The visualization of multiple types of graphics (mesh, contours and vectors) is now simplified thanks to the scenes definitions.

Fluent Meshing

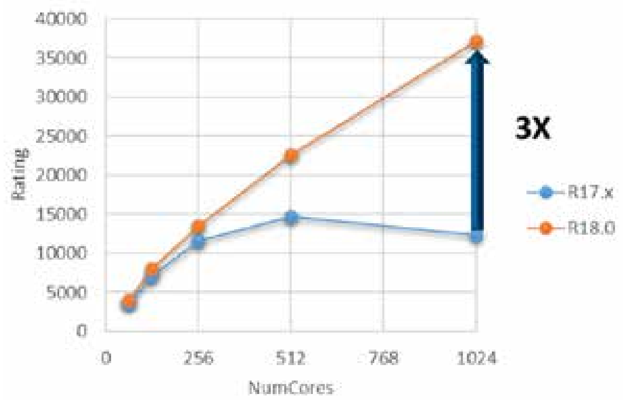
A CAD assembly mode is used to represent the CAD tree in the same way it is presented in the CAD package it was created with all sub-assembly levels. The user benefits from improvement in prism generation (stair-step handling), Octree HexCore methodology for cell creation and over 100 new utility functions to aid batch executions. The



new application AdvWrap ACT can be used for automated pre-process using Fluent Meshing with support for wrapping and volume meshing on complex industrial models. The Fluent Meshing technology is now accessible in the workbench environment with a dedicated application, Workbench Poly Meshing.

HPC

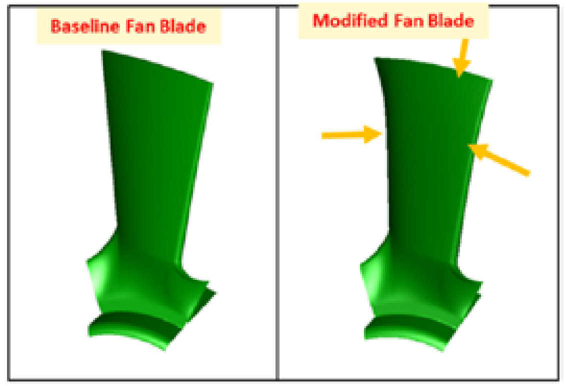
The node-based gradients are now optimized for better performance in the external aerodynamics. Large models with sliding interfaces are now close to 3x faster thanks to the scalability improvements. R18 has massive write performance gains for HDF-format files (14x faster), extensively used for parallel computations.



Physics

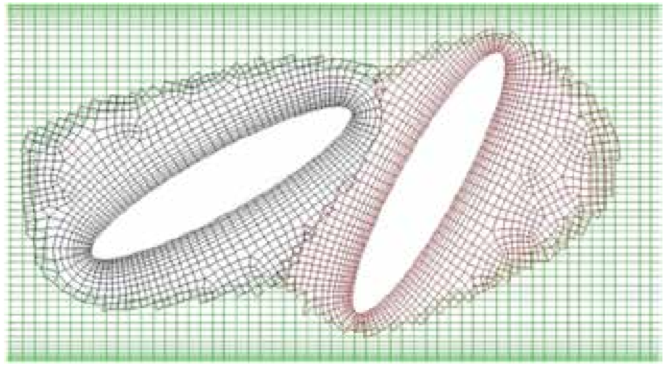
Different improvements aid convergence and robustness for multiphase and reacting flows, heat transfer and radiation, for the all the typical industrial cases.

New model accurately simulates water-oil displacement in reservoir production, with the Corey model for 2-phase relative permeability in porous regions. Multiple compressible phases including arbitrary



combinations of ideal-gas and real-gas can be used for high pressure injectors. The distortion of bubbles in Eulerian flows can be included with new drag laws derived from ANSYS CFX. The injection properties for spray can be defined as input parameters and the cone injection has been unified including different sub-types. A new binary condensation wall film condition is available for Lagrangian simulations.

The visualization of 4-dimensional or 5-dimensional PDF tables can be plotted using 3D surface or 2D curves. The finite rate chemistry workflow has been improved with Chemkin-CFD solver which is now up to 21x faster.



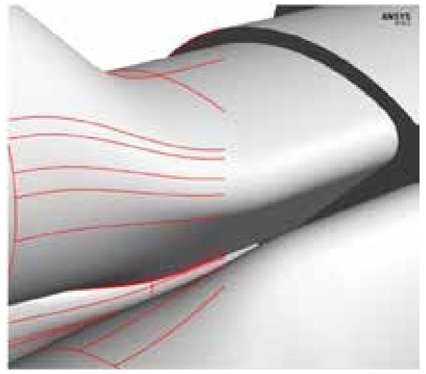
The Monte Carlo radiation model is fully implemented and well-suited for collimated radiation problems where DO is too expansive, ie. headlamps.

The adjoint solver supports rotating machinery with the possibility to add cylindrical deformation regions; a new suppression option is available for dissipation stabilization scheme.

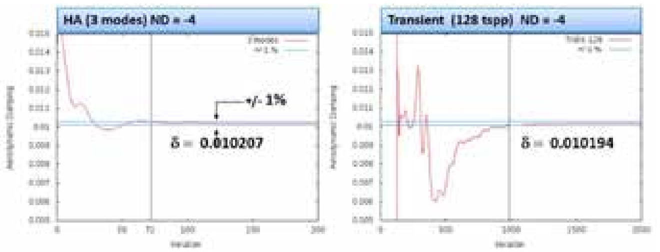
Modeling flow involving interaction bodies with arbitrary motion can be approached now with the overset mesh technology. Overset mesh can simplify and speed simulations that include mainly moving cell zones without having to use re-meshing or smoothing. The generation of the mesh for complex geometries is simplified and the grid quality is maintained during the motion. A user interface similar to non-conformal interfaces can be used with minimal user input. This technique is now compatible with pressure and density based solvers, laminar and turbulent flows (k-epsilon and k-omega), heat transfer, volume of fluid, dynamic and sliding meshes; all cell and mesh types are supported.

ANSYS CFX R18

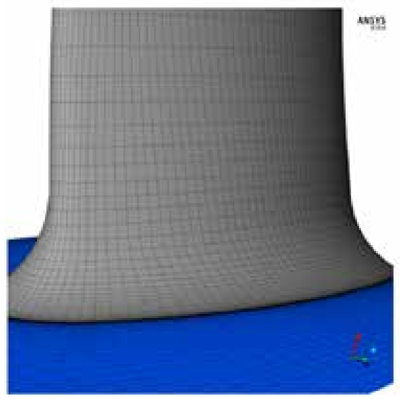
ANSYS CFX is the flagship product for turbomachinery CFD simulations and is recognized for its outstanding accuracy, robustness and speed. New features and enhancements are described below.



HPC
The user can now turn on face set topology simplification to massively improve solver performance for models with a large number of 2D primitives. With this new option it is possible to avoid unnecessary internal loops while preserving 2D regions for post-processing, gaining a speed-up of 50% for some cases. There is a new option for pre-coarsening meshes before partitioning takes place. This option (AMG coarsening) improves efficiency and robustness, and helps to avoid convergence difficulties, particularly with diffusion-only type equations, by helping to prevent partition boundaries from passing through areas of high aspect ratio cells. The solver now reads and sends compressed data; this allows a general improvement in file writing performance and a sensible speedup of the file reading and writing in parallel runs with large core counts.



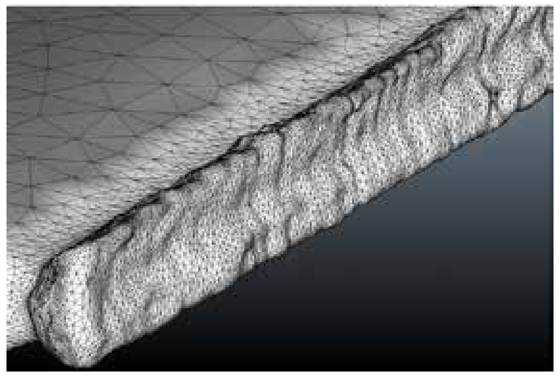
Transient Blade Row
Transient periodic flow in blade flutter cases can be solved now more efficiently by using Harmonic Analysis (HA), a frequency-based solution method that avoids the need to march in time. Validation cases show a speed-up from 10x to 100x respect to transient periodic flow with a classical transient time integration.



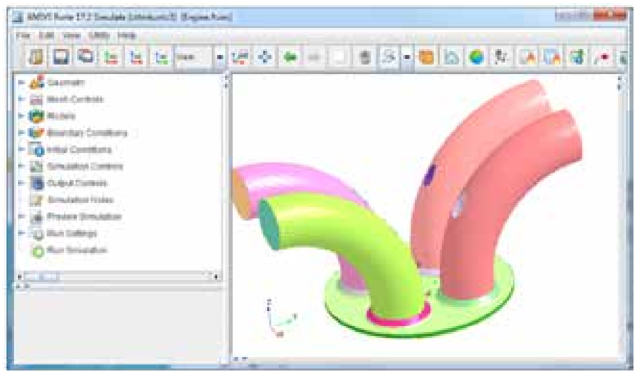
TurboSystem - TurboGrid
With R18 is it now possible to import CAD models directly into TurboGrid, with an exact representation of the blade geometry. This new feature allows the user to improve quality, fidelity and robustness in the design process. There are 2 possible solutions:

- transfer via workbench schematic: ANSYS BladeEditor exports underlying, native blade surfaces from and meridional flowpath in Parasolids format
- direct import: supports Parasolids (x_b, x_t) and ICEM CFD Tetin (.tin)

ANSYS FENSAP-ICE R18
FENSAP-ICE is the premier in-flight icing simulation system, used worldwide by major aerospace companies. Turbo module analyzes the external flow and icing over the aircraft, with engines installed and running. It has no geometric limitations so you can apply its results to all internal and external components of the jet engine. With the new release, the tools is completed included in ANSYS with a Workbench plug-in, connections to Fluent/CFX, results analysis in CFD-Post and parameterization.



ANSYS Forte R18
ANSYS Forte is a tool dedicated to the simulation of combustion in IC engine with nearly any fuel and helps engineers to design cleaner burning and high-efficiency engines. An automatic on the fly mesh generation and an innovative solution adaptive technique are coupled with robust and accurate combustion modeling capabilities. R18 is characterized by a significant advance in spray modeling with reduced mesh dependency and faster turn-around time, especially for GDI cases.



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

ANSYS 18: Composite Cure Simulation

The ANSYS 18 release confirms the ANSYS vision to enable the customers to reach lighter, stronger and more efficient products. The today's main industrial challenges are strongly dependent to the fundamental transformation in manufacturing processes. Looking at high performance materials, composites provide new possibilities for designers and manufacturers allowing stronger, lighter and more cost-effective solutions, but at the same time, they pose new modeling and manufacturing challenges because of the nature of the materials. With the right simulation tools, designers can account for residual stresses, predict performance, analyze reliability and potential failures, optimize construction, and export accurate information to manufacturing division, all before a physical prototype is built. In particular, in ANSYS 18, a tool is available to analyze composite curing process ensuring results reliability and accuracy: it is ANSYS Composite Cure Simulation (ACCS) module, fully integrated into ANSYS Workbench platform. The numeric simulation of curing processes in composites helps to fulfill the increasing requirement for small component tolerances and fewer rejections of defective parts, and finally also improves the part performance.

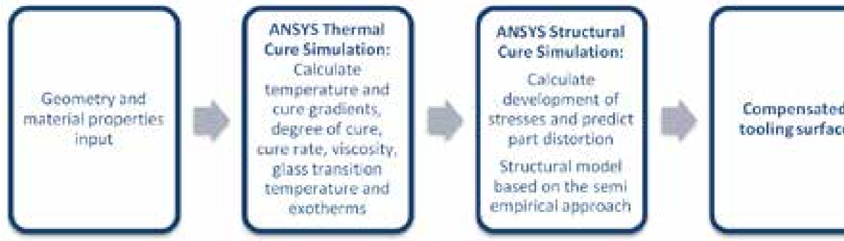


Figure 2 – ACCS Analysis steps

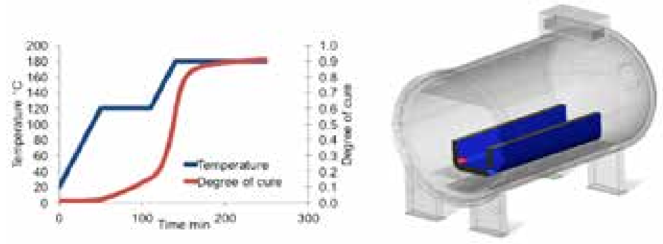


Figure 3 – Degree of cure evolution during curing process

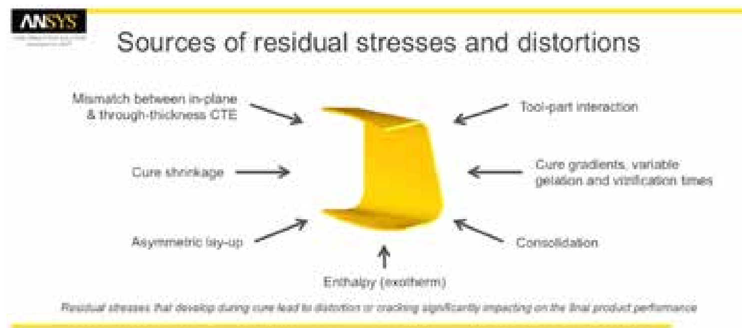


Figure 1 – Sources of residual stresses and distortions

ANSYS Composite Cure Simulation Highlight
ACCS allows to simulate composite curing process using thermal-structural analyses predicting composite properties evolution and distortion and residual stresses at the end of the curing process. These outputs allow designers to compensate tooling surface reducing final distortions and reaching desired shape.

In details, ACCS chemical solver is embedded within the transient thermal module and simulates development of polymerization and thermal history. This allows the analyst to verify the curing process itself. As example, it is possible to check if the composite component is fully cured of if it is necessary to optimize the temperature profile of the autoclave process to avoid unfairly cured zones or detrimental exothermic peaks.

Subsequently thermal and cure data are passed into the structural data module where ACCS structural solver calculates development of residual stresses and process induced distortions.

This novel approach, coupled with optimization analyses, allows to produce composite components without costly trial and error methods, significantly shortening time to market as well as the overall product and assembly cost.

For more details:
Alessandro Mellone, EnginSoft
a.mellone@enginsoft.com

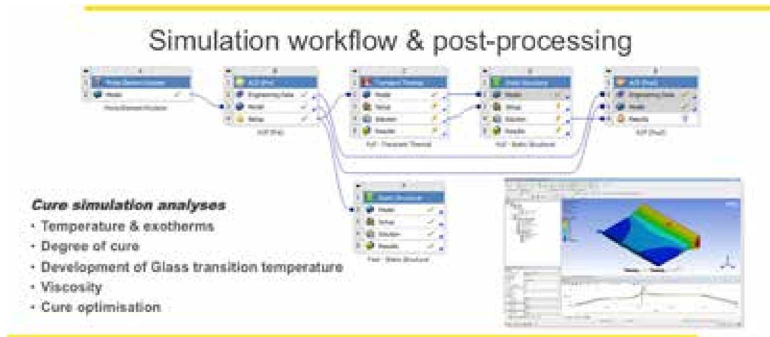


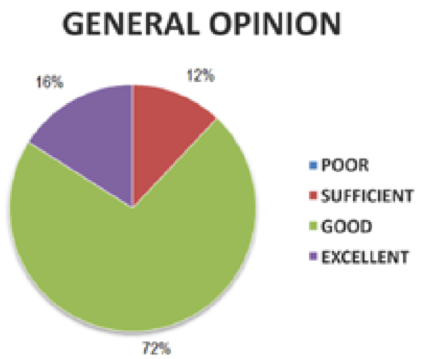
Figure 4 – ACCS simulation workflow

FOUNDRY 4.0.
The foundry digitalization. Process Control and the costs of non-quality

On March 23rd a Training Day on Foundry 4.0 was organized by the Diecasting Technical Committee of the Italian Association of Metallurgy. The event was held at the Kilometro Rosso Science and Technology Park in Bergamo, and gathered about 70 interested participants.

The context

Following the Industry 4.0 project in Germany, Italy has reacted with enthusiasm and concreteness as proved by several individual companies first and then by the recent adoption of the National Plan Industry 4.0 promoted by the Italian Government. Enabling Technologies are well known in many industrial fields where the process digitization has always been a fundamental prerequisite for high virtualization and automation. Diecasting, among all foundry processes, represents the most suitable process, despite its complexity, for an intensive monitoring the process parameters. Many parameters are already under control today but the various "actors" constituting the diecasting island are often managed by different control panels, while the process complexity would require an intelligent management of all the variables concurring to definition of the casting quality, the energy consumption and the final product cost.



If on one hand production data and the related performance are monitored by management systems to assess the EEO reference values, several information comes from the core mechanisms of the productive cell; today is possible to equip with sensors and monitor every single stage of the process. These big amount of data does not constitute a knowledge and an advantage if the existing reference correlations not extracted and processed in real time; nowadays cognitive systems are able to predict the quality of every single casting.

The maturity and applicability of these new technologies in the age of digitization has been discussed by representatives of different business entities to understand how to transform the Foundry in a factory of the future, to be really called Foundry 4.0.

The event

The day was divided into two parts: a first technical session, addressed to the production and quality managers, technical staff and IT experts managing the process data, and an afternoon session focused on the debate for CEO, CTO, CFO and Managers to explain how a digitization project can be implemented in their company by assessing the possible objectives, the necessary investments and the available financial instruments.

As in other industrial contexts, also the foundry can take advantage of the Industry 4.0 plan, evolving in the digitization and integration between man and production line. All agree that in the supply chain, foundry plays a key role and control panels have been used to acquire data for a long time; therefore the already existing production lines are not excluded and the operability and data management have to evolve into new intelligence forms and knowledge models. In order to avoid the acquisition of disaggregated data from the process by the different devices, it is advisable to look for standardization, centralization and consistency on the indices that define production efficiency and stability, as well as the overall casting quality. The knowledge and data value have to be extracted so to be reused with appropriate corrective actions.

The addressed topics, as listed below, have been introduced by a presentation concerning the diecasting sector in Italy, its potentialities and current competitiveness driven by innovation.

The general opinion is reassuring and quite good.

Technical session:

- The production data management with MES tools and related case study by an industrial speaker.
- The innovative solutions related to the press for plant control.
- The new technologies to monitor the mold and their application during the design phase of the equipment.
- Defects classification and quality and cost models.
- Innovation on high pressure die casting process control and application of the quality predictive models in real time with and the application case presented by industrial speaker.

Session debate:

- Financial resources and instruments are to support the Foundry 4.0 plan.
- The foundry needs aiming at improving the OEE in the industrial field.
- Example of digitization project and traceability in the foundry.

Software Update

The debate session was enriched by a round table, fostering a constructive discussion among the participants and the companies that joined the event. The key factors contributing to project are traced to three areas: human resources, where new roles and competences are required; enabling technologies 4.0, that help the collection of data by integrating the operator with the production line and the financial instruments.

In particular

EnginSoft played a key role in this event, with particular reference to the following two relevant contributions:

High pressure die casting process control and quality predictive models in real time
Nicola Gramegna - EnginSoft

The production efficiency is supported by the centralization of all the data of the production line, that allow to monitor and correlate the different phases of the production process, from the alloy and injection, to solidification and cutting, to predict and therefore optimize the casting quality in real time so to react accordingly when necessary. The sophisticated quality predictive model is applied in line to immediately detect scraps during the entire production.

There are convincing reasons leading to digital transformation, which have been carefully analyzed, drawing to the conclusion that quality and efficiency impacts can be pursues using the new platform "Smart Prod ACTIVE"

1. Remote control of complex multiple stages processes (e.g. Foundry) based on different tools
• Process stability; minimum machine failure
• Application also to traditional processes (e.g. Foundry)
2. Improvement in production efficiency (OEE) with reference to Measure => Analysis => Action
• Speed up the process of setup (optimization)
• Real time adjustment of process parameters
• Toward "zero defects" with quality prediction at 100%
3. Increase in knowledge of the process within data (understanding the data value)
• Use of new knowledge in flexible production (machine learning)
4. Sustainability and Profit
• Reduction of energy consumption
• Reduced time-to-market
• Minimization of costs

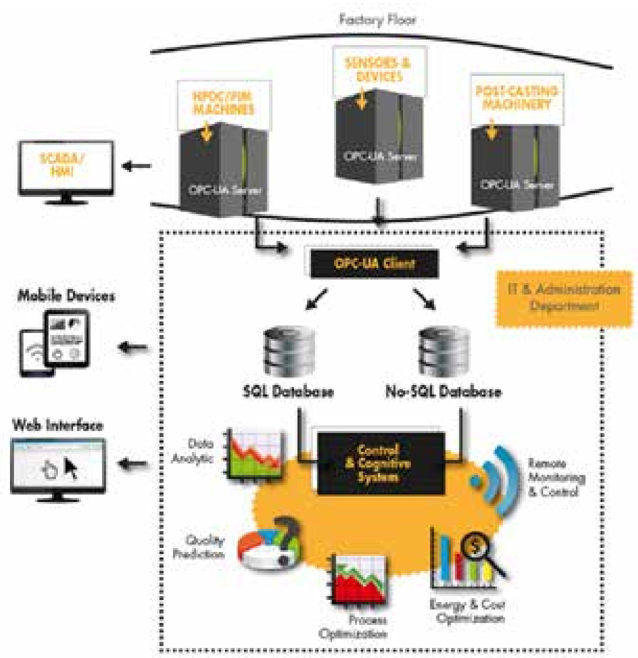
FOUNDRY 4.0 - Factory of the future
Angelo Messina - EnginSoft

Introduction to the concept of the digitalization of information and related advantages.

Presentation on how this innovation can be collocated within the manufacturing sector and even applied to the most traditional processes.

Enabling technologies and tools of the revolution Industry 4.0. Indications about how to face a digitization project and take advantage of the funding introduced by the Industry 4.0 Plan.

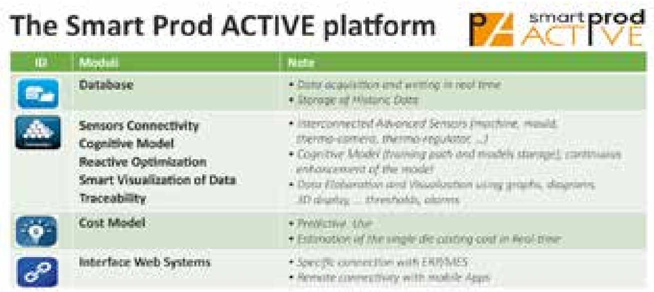
The National Plan for Industry 4.0 considers different and consistent incentives for investment in equipment and technologies 4.0: Hyper



depreciation (250%), Super Depreciation (140%), New Law Sabatini, R&D. These incentives are mostly cumulative, making this investment even more attractive..

It is important to understand that, despite the incentives promoted by the Government, the transformation toward Industry 4.0 does not only mean making an investment in machines. Smart Manufacturing does not mean adopting a single technology for a given process, but it really refers to a an overall the digital innovation, able to reinvent the mechanism of value creation inside a company (plants, people, information). In this perspective it is essential to rethink and reinvent both the managerial technical skills of the human resources.

For more information: Nicola Gramegna - EnginSoft
n.gramegna@enginsoft.com



ID	Moduli	Note
	Database	• Data acquisition and writing in real time • Storage of Historic Data
	Sensors Connectivity Cognitive Model Reactive Optimization Smart Visualization of Data Traceability	• Interconnected Advanced Sensors (machine, mould, thermo-camera, thermo-regulator, ...) • Cognitive Model (training path and models storage), continuous enhancement of the model • Data Elaboration and Visualization using graphs, diagrams, 3D display, ... thresholds, alarms
	Cost Model	• Predictive Use • Estimation of the single die casting cost in Real time
	Interface Web Systems	• Specific connection with ERP/MES • Remote connectivity with mobile Apps

MAGMA 5.4 "Revolution" at METEF

The castings are getting more and more complex in shape, with high performance requirements, and at the same time the robustness of process and high quality; in other words, the casting design requires a compromise of acceptable cost, weight reduction, appropriated mechanical properties, structural integrity and stable process.
In the design phase of the whole process, considering each phases that characterize it, the optimization and control allow to answer effectively to the increasingly stringent demands of the markets.
To achieve these objectives, the Design Chain approach is increasingly applied to evaluate the best solutions in terms of castability and mechanical behaviour.

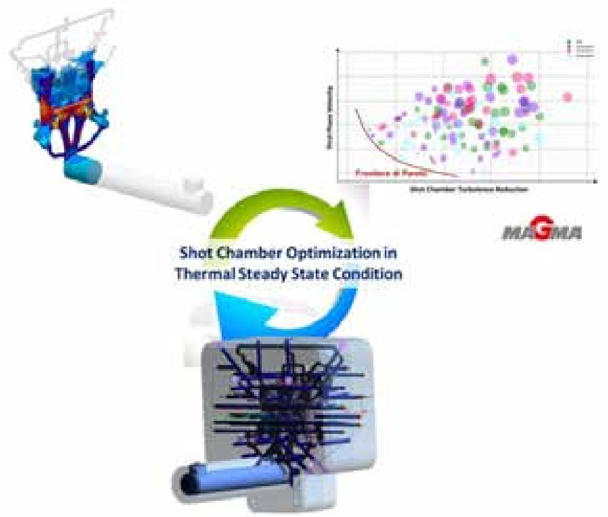


Fig 1- Shot Chamber Optimization (Chamber dimension, plunger velocity, chamber thermoregulation to avoid waves and distortion and to increase the alloy quality)

One of the most relevant and Virtual successful innovations on recent years is the Automatic Process Optimization to satisfy all the casting requirements.
The MAGMA 5 Optimization module is evolving to support the design during the research of the best engineering solution.
The latest development, MAGMA5 Rel. 5.4, allows, moreover, the complete optimization of the entire process considering for example the chamber, the effect of lubrication and the thermo-regulating effect of the cooling-chanel designed.
Automatic optimization aims at calculating and identifying optimized casting process variants. For this purpose, the software automatically varies the rigging system within the bounds of the predefined degrees of freedom. Thus, in contrast to virtual designs of experiments, most experiments are yet unknown at the beginning of the optimization run.
On the basis of given objective functions (and secondary conditions where applicable), the quality of each calculated variant is then evaluated. The automatic variation of the variables defined is initiated based on a given Design of Experiment sequence (start sequence). Depending on the achievement level of the objective function for each experiment separately, new variants are created, simulated and evaluated again. Inspired by the genetic processes in biology, this approach comprises several generations and uses mechanisms of inheritance, mutation and selection.
For Automatic Optimization with MAGMA5 Rel. 5.4, the user does not need a thorough knowledge of the optimization method, since the software automatically performs the optimization itself. The user is provided with the tools for the analysis and the detailed assessment of the virtual designs of experiments carried out.
The new release Magma 5.4 introduces the user to the new Optimization features including: the injection chamber (Fig. 1), the lubrication advanced models (Fig. 2), the thermoregulation advanced models and the model of ejection (during the cast extraction)
These new features will be integrated with the Optimizer already present with the previous version 5.3 and integrated in the standard version of the software.
It will also be possible to maximize the calculation time economy thanks to the new mesh and calculation solvers.
Magma 5.4 is available for 64-bit operating systems as Windows (Win7 and Win 10) and Linux (Red Hat Enterprise 7, Suse Enterprise 12).

We invite you to visit us from 21 to 24 June at the Metef Trade Fair, in Verona, at our booth (D14/E15 - Pad. 4), where you can take a look at all the news of the software.

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

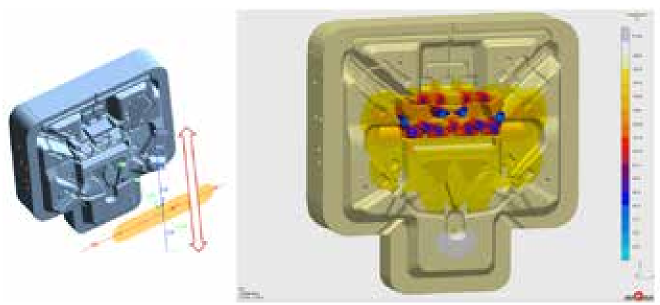
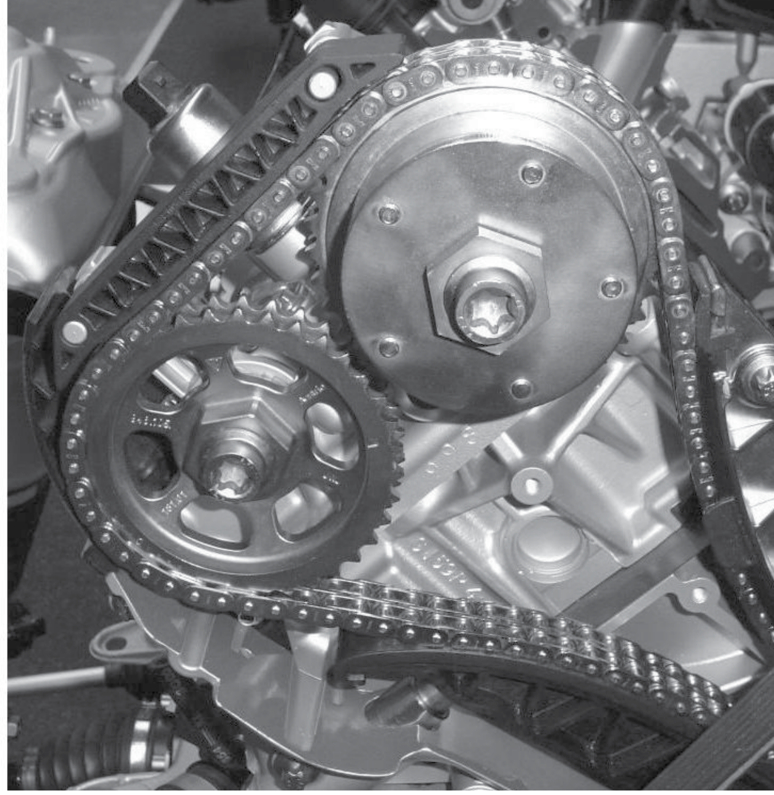


Fig 2 - Lubrication advanced models: movement control, lubricant pressure control, nozzles distance and type, lubricant type



Design and Analysis of Chain Drives made easy by RecurDyn Multi-Body Software

RecurDyn is an emerging simulation technology to investigate both kinematics and dynamics of moving mechanisms. Despite being relatively young, RecurDyn is gaining significant portions of the Multi-Body-Simulation market by exploiting its unique features. In particular, the platform and its solver are optimized to efficiently manage virtual models characterized by high frequency dynamics and/or highly discontinuous phenomena. This is the typical scenario when the mechanism includes flexible bodies and/or contacts. Flexibilities bring into the model high frequency vibrations that combine with low frequency motions, whereas contacts add repetitive impulses that disturb the solution and excites vibrations.

Figure 1 – Polybolos reconstruction by the German engineer Erwin Schramm (1856–1935)

The power of RecurDyn's solver is really made available to the users through an intuitive GUI, that speeds up any pre-processing task that requires hand work. In addition, the platform includes a series of so called "toolkits" which automate the modelling of typical mechanical subsystems such as gears, belts, bearings, springs, chains, and more. Each toolkit creates bodies, joints, contacts and forces in accordance to the user inputs and, at the same time, adds special analytics for the system that is created.
Although RecurDyn can be used effectively in a unlimited number of situations, one of the applications where it really makes the difference is the simulation of chain drives. This short article presents the RecurDyn chain toolkit, which makes possible to model, simulate and analyze several types of standard chain transmissions. The customized interface of the toolkit guides the user through the necessary steps to assembly sprockets, links, guides, and tensioners. The internal library includes ISO standard components, so that even the CAD is automatically created. From this short introduction, the reader might argue that non-standard chains cannot be analyzed with RecurDyn. This would be a hasty conclusion. Indeed, RecurDyn features a programming environment, called PNet, through which the user can conveniently automate the operations that are needed to adjust the

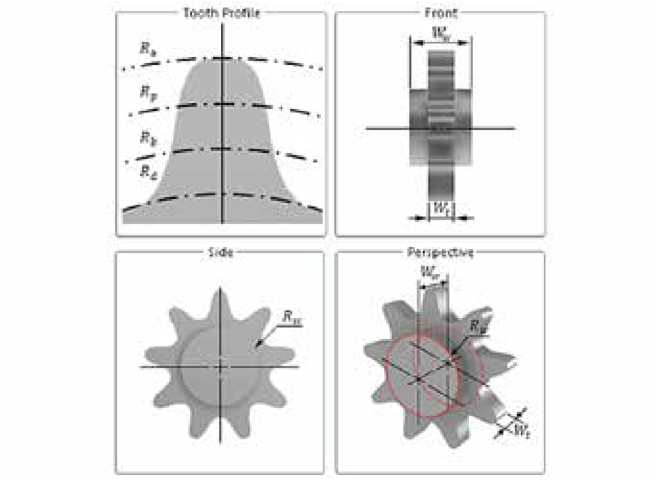


Figure 2 – Sprocket properties

model towards a fully customized chain.
Chain drives are mechanisms used to either transfer power from one location to another or to carry objects around. The first documented chain drive dates back to the 3rd century before Christ, and was used in a polybolos (Figure 1), which is a very ancient missile weapon. Chain drives have been improved a lot since then and cover a variety of application in any field of industry today. Besides being relatively cheap, they have a large power-to-weight ratio, they guarantee the position of the driven parts and can be used to cover a large range of distances. The use of chains is so diffused that is even difficult to select a main application as reference. In this article we will focus on the analysis of a simple timing chain, that is used to synchronize the camshaft rotation with the crankshaft rotation in internal combustion engines. This chain transmission has severe requirements such as precise positioning, low noise emission, large range of operating speeds, long time resistance, and so on.

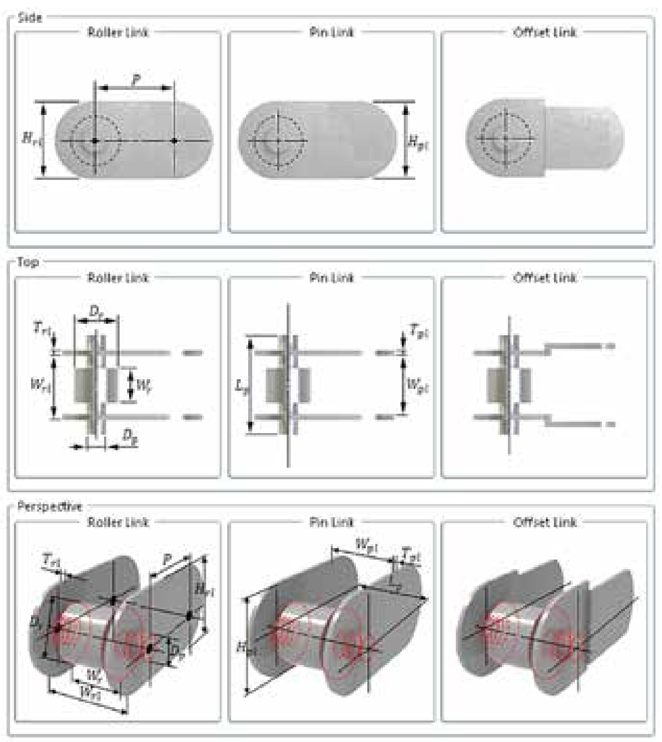


Figure 3 – Link geometrical properties (roller chain type)

Model creation through RecurDyn/Chain Toolkit

Once the Chain Toolkit is activated from the GUI, a set of specific icons appears in the menu bar. The toolkit is designed to model a chain drive composed of rigid bodies only. The CAD of these bodies is internally generated as the user inputs the necessary properties. The sprockets are the toothed gears that provide both input motion and any resistant torque in the chain drive system. Each sprocket requires the input data shown in Figure 2. The chain drive might also feature also some rollers, which are like pulleys that affect the chain direction without transferring any longitudinal force.

Once the rotating elements are all in place, then the user is requested to define the chain itself. This task is accomplished by first choosing from the menu the type of link. The user has to input the sizes of two links only, since the chain is just a series of them. RecurDyn makes possible the modelling of roller chains, multiplex chains, and silent chains. As an example, the Figure 3 shows the input data to completely define the links of a roller chain type.
The third crucial task, fully automated in RecurDyn, is the assembly of the chain. The user is requested to trace in sequence the straight branches of the chain path, by simply touching with the mouse the sprockets and the rollers that have been previously built. Once done with the path, the software replicates the links so that the chain completely covers the path. The resulting chain is fully connected and approximately engages the sprockets. In a timing chain the path is clearly closed (chain loop), but other applications might require an open path; RecurDyn can manage both.
Further components, which are very common in a chain drive system, are the guides and the tensioners. The guides are either curved or straight pads that prevents the chain from radial and/or lateral motion. Sometimes guides are not fixed to the chain bench and are coupled with a piston that provides a controlled force. This device is known as tensioner and is really important to keep the desired level of longitudinal load as the chain elastically deforms or gets longer for wearing. A simple timing chain looks like the one shown in Figure 4. In this example both driving and driven sprockets are attached to the workbench through revolute joints. The driving joint is coupled to a rotational actuator, whose angular speed matches a time history measured in laboratory. The driven joint is coupled to a rotational actuator, whose resistant moment matches the torque signal measured in laboratory as well. Therefore, the boundary conditions for the chain system are reliably reconstructed. For a more detailed analysis, the timing chain model could be extended by adding the crankshaft, the valve shaft, and more components.
In this demonstrative and basic model, the tensioner force has been modeled as simple elastic reaction, represented with a spring icon in Figure 4. More generally, RecurDyn makes possible to define much more sophisticated models of this force, including hydraulics, non-linear friction, and non-linear springs. Moreover, the dynamic model can be connected to matlab Simulink or AMEsim, when more sophisticated models of the tensioner force are available in these platforms.

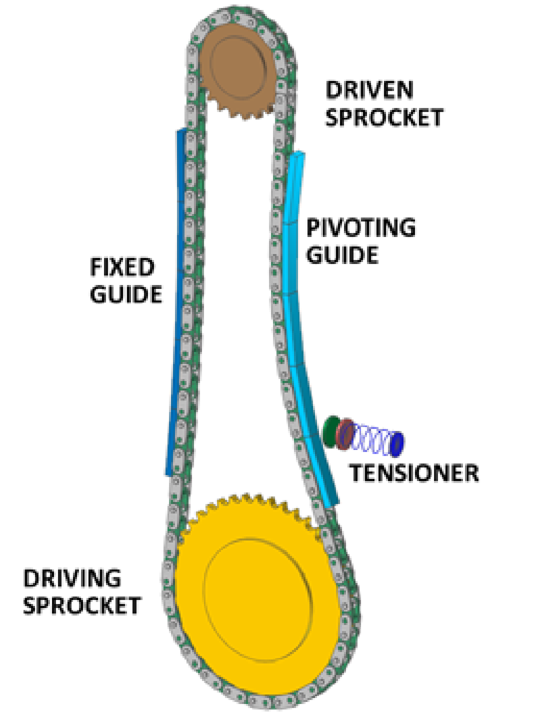


Figure 4 – Multi Body model of a timing chain

Chain Model Characteristics

The chain toolkit assemblies the model and automatically creates the contacts and the joints through which all parts interact together. Chain links are held together by bushing elements, which are 3-dimensional force elements. Each bushing applies 3 relative forces and 3 relative torques between the connected elements, which are calculated as functions of relative displacements, relative rotations, relative translational speeds, and relative rotational velocities. Stiffness and damping functions can be fully customized, so that any type of global response can be matched. As an example, by setting force and torque functions with a flat dead zone in the zero displacement range, it becomes possible to model the effects of gaps between the links. In general, the bushing properties must be calibrated to match the experimental response of known chains. It is worth to point out that deformable bushings at link-to-link connections make possible to describe the chain elasticity that, otherwise, would not be available with rigid bodies. The so modelled chain interacts with sprockets, guides and rollers via non-linear frictional contacts. RecurDyn internally recognizes the shapes of the contacting surfaces, so that there is no need for the user to select them manually. Contact parameters such as stiffness and damping need to be input by the user, in accordance to FE analyses or experimental measurements. Depending on the type of chain.

Simulation and Review of Results

Chain models are among the largest and most complex that can be simulated with multi-body technology. First, depending on the chain length, the number of moving bodies could become remarkable. Second, contacts disturb the solution all over the chain, causing continuous impulsive excitation of the mechanism. Third, chain links are bodies with reduced inertia, whose motion is driven by stiff contacts and stiff bushings. RecurDyn has built its reputation on chain application. The main reason is its hybrid solver, which uses an innovative approach with respect to its competitor. Even chain models featuring hundreds bodies and thousands contact points can be simulated in a reasonable time.
Once the simulation is accomplished successfully, RecurDyn offers the possibility to extract and post-process a bunch of output quantities. By default, the model returns both translational and rotational quantities of each part, such as position, speed and acceleration in space. Simulations of very long chains generate large output files that are difficult to handle. For this reason, the user can select in advance the links he is interested to investigate in detail. The RecurDyn model also outputs both forces and moments on each link-to-link connection, as well as the contact forces between chain and any other element it interacts with. Since the chain dynamics is quite discontinuous, time histories of selected output quantities might look rather meaningless (i.e. high frequency oscillations). That's why RecurDyn post-processor includes specific tools such as frequency filters and FFT, that really facilitate the user's comprehension. More deeper investigations can then be performed outside RecurDyn using the data that are easily exported in txt format.
RecurDyn, in combination with its Chain Toolkit, is really the software solution for the industries that design chains or systems including chains.

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

MathOnJob

The last edition of "MathOnJob" was held last May 24, 2017 at the Department of Mathematics of the University of Bari "A.Moro". The event was attended by the most important multinational firms including GPI, OmnitechIT, Ernst & Young and many more. Among these, EnginSoft had the chance to highlight its leadership in the field of production processes and virtual prototyping for the most important national and international companies, due to its experience of over than 30 years.

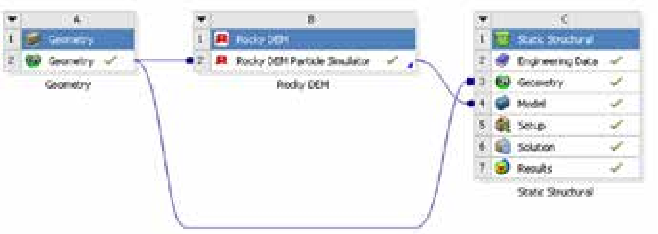
Representing EnginSoft was Marco Perillo, Technical Director of EMT Mesagne, who presented the company and illustrated the training opportunities offered by the company to tesists and graduates, along with Elio De Marinis, mathematician and developer, who showed how mathematics can play a key role in resolving industrial issues also through software distributed by EnginSoft, such as Maple and MapleSim. In particular, a number of regional and European projects leveraging mathematical modeling as well as numerical tools developed in collaboration with polytechnics and universities, were exhibited.

There were numerous questions asked by students not only about the training aspects and career opportunities offered by EnginSoft, but also aimed at better understanding the various contexts in which the mathematicians could be involved from the company.

For more information: Elio De Marinis - EnginSoft
e.demarinis@enginsoft.com

Multidisciplinary simulations with ROCKY

As for many other industrial sectors, also the processes involving bulk materials are requesting more efficient and productive machineries and plants to compete globally.
Designers need to use equally efficient and productive tools to achieve more advanced and complex designs. In order to reach these results, the whole system has to be analyzed, taking into account also the interaction between different physics, and therefore, performing multidisciplinary simulations. This is precisely the context in which the combination of ROCKY and ANSYS creates a single tool able to provide designers with everything they need to achieve these objectives.



ROCKY
It is a 3D discrete elements software (DEM), leader in the sector of granular materials simulation. ROCKY allows to accurately analyze the flow of such materials inside the whole process of transport and processing, by representing particles with their realistic form. The solver uses the parallel computation both on CPU and GPU. This last powerful and innovative feature allows to highly improve the calculation performance and opens the way to the resolution of problems with a large number of particles by quickly providing results to be integrated within the industrial design process.

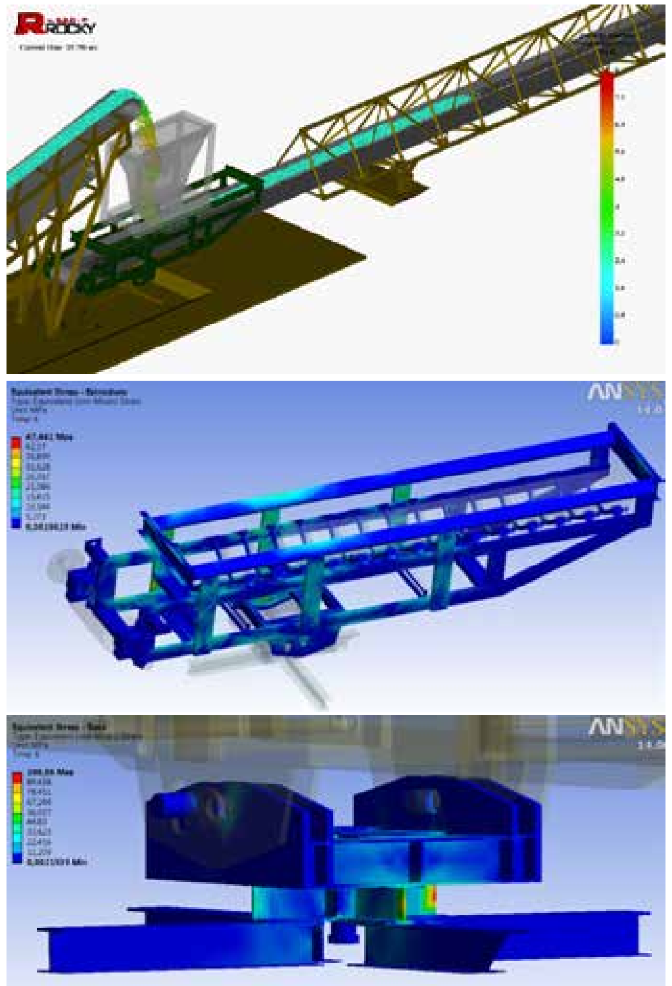
ANSYS
It is a world leader in the CAE sector. The Workbench suite enables its users to efficiently and quickly perform complex simulations in the mechanical, fluid-dynamic and electronic sectors.

ROCKY & ANSYS
ROCKY is an independent software fully integrated within the ANSYS WORKBENCH suite. This important aspect allows to couple the simulations carried out with the DEM solver with other physics, as fluid-dynamic or structural analyses. The integration of these two software allows to work within a single environment and to exchange information between the different solvers in an efficient and correct way, thus drastically reducing the models preparation time.

ROCKY and ANSYS MECHANICAL
The coupling in the mechanical sector allows to transfer the granular material loads on the plant and therefore to evaluate stress and reactions on the single components of the system, highlighting any possible structural criticality. The information transfer from ROCKY to ANSYS MECHANICAL can be performed by connecting the DEM results with static, dynamic and fatigue analyses.

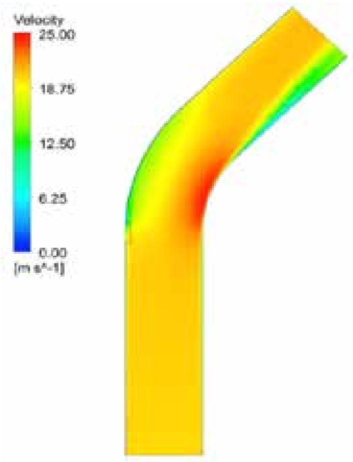
ROCKY and ANSYS FLUENT
The DEM-CFD (ROCKY/FLUENT) approach turns out to be crucial any time an assessment concerning the interaction effects between solid particles and fluids has to be performed, in order to improve the process under examination or the tools characterizing the process itself. Some important real cases to estimate such effects are mainly represented by: air/water separators/conveyors, systems for granular material fluidization, sewage treatment systems or granulates dryers.

ROCKY and FLUENT can be coupled in two different ways: 1-way and 2-ways method.
The first and easier method allows to apply the fluid dynamic data to the particles, permitting an evaluation of the fluid motion influence on the granular material dynamics. This approach can be applied if the particles motion effects on the fluid itself can be neglected.
With the 2-way method the particles are an integral part of the fluid and therefore it is no longer possible to ignore the motion effect inside the fluid itself. How is information exchanged?
On the ROCKY side, the fluid exerts its force on the particles due to the pressure gradients (including the buoyancy effect), the frictional forces and the added mass. On the FLUENT side, the reaction of the forces upon the particles will be applied over the fluid phases. The two solvers run in parallel, exchanging information at each iteration. For this kind of simulation, the ROCKY parallel GPU solver, is essential. This important feature allows the users to run ROCKY completely on graphic cards without subtracting processors to the FLUENT solver and therefore obtaining an higher computational efficiency.

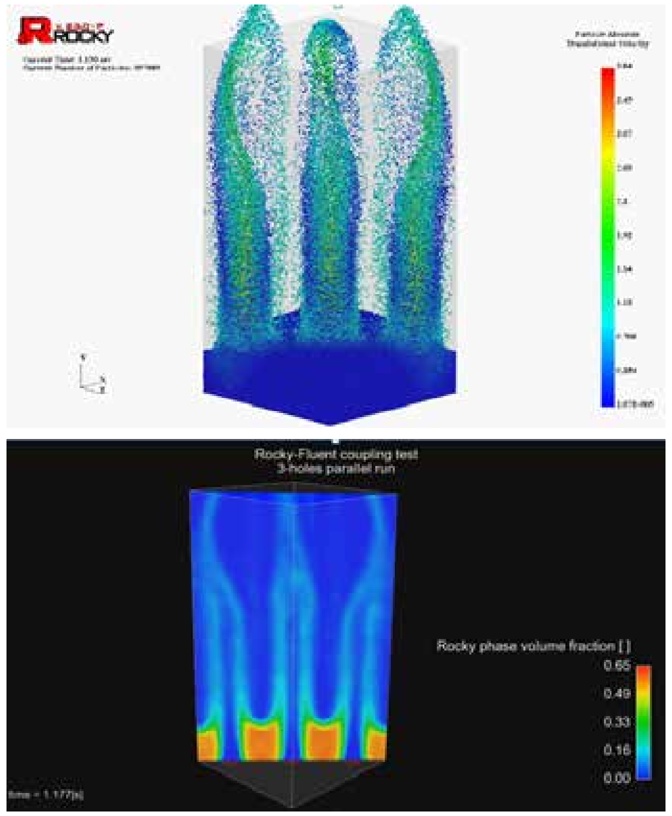


ROCKY – ANSYS WB Mechanical

M. Tomasi - EnginSoft
For more information:
m.tomasi@enginsoft.com



1-Way Coupling FLUENT- ROCKY Windshifter



1-Way Coupling FLUENT- ROCKY - Windshifter

Free Maple Player

If you develop a new application in Maple, and you want to share it to your colleagues without having to buy additional licenses and/or modify the document, the Free Maple Player is the correct choice! An example of application that you can develop and share is reported in Figure 1.
The Maple Player is available for Windows, Mac, and Linux . There's also a downloadable Maple Player for IPad to explore mathematical concepts and solve advanced problems on your IPad (Figure 2)!
The Maple Player is a free application that lets you view and even interact with Maple documents to solve problems, visualize solutions, and explore concepts, even if you don't have Maple. Use the Maple Player to view documents from colleagues or sample the vast collection of Maple documents already available from Maplesoft and the Maple community in the Application Center, the Teacher Resource Center, Maple Primes, MapleCloud, and more. Within technical organizations, use the Maple Player as a free deployment option for sharing Maple-based solutions internally.

With the free Maple Player, you can:
- Use interactive Maple applications to perform computations and visualize results.
- View any Maple document, regardless of author or source.
- Easily access the collection of interactive Math Apps available through built-in access to the MapleCloud.

Perform any operation that uses interactive components, including entering mathematical expressions, moving sliders, playing videos, and rotating plots, with limitations only on the number of characters that can be entered.

For more information:
Manolo Venturin
m.venturin@enginsoft.com


Figure 2 - Example of the Free Maple Player on an Ipad(R). Source: http://www.maplesoft.com/products/mapleplayer/


Figure 1- Example of an GUI interface that you can develop and share with Free Maple Player

GENESIS Structural Optimization for ANSYS Mechanical

New Features and Enhancements

GENESIS Structural Optimization for ANSYS Mechanical (GSAM) is an integrated extension that adds structural optimization to the ANSYS environment. The extension provides an easy-to-use interface which allows the user to setup structural optimization problems, post-process them and export the optimization results within the ANSYS environment. GSAM is a super set of GTAM (GENESIS Topology Optimization for ANSYS Mechanical). GSAM can perform any function that GTAM does. The extra functionality is to perform topography, freeform, sizing and topometry design.

The new version of GENESIS 16.0 has several new enhancements to improve its usability. For example, the topology optimization module in GENESIS has been changed to handle new responses such as temperature and heat transfer compliance. New fabrication constraints have been added to this module; among them are: minimum thickness filling. GENESIS can now solve multi-model optimization problems, where different analysis models need to share common designed regions.

Structural Optimization Enhancement

- Topology Optimization for Heat Transfer Analysis: Heat transfer analysis now is supported for topology optimization. Temperature response can be used for topology optimization with heat transfer analysis. An additional new response, Heat Transfer Compliance (HTC), is added for heat transfer load case.
- Heat Transfer Compliance Response: In thermal load case, the heat transfer compliance (HTC) is defined as HTC= 1/2 $\{T\}^T \{F\}$, where {T} is the vector of grid temperatures, and {F} is the vector of applied heat fluxes. When HTC is minimized, temperature at grids where heat fluxes are applied is minimized.
- Strain Response: Strain response are now available through GSAM/GTAM interface.
- Grid Stress: Grid stress and dynamic grid stress are now available through GSAM/GTAM interface.
- Updated DOT: The latest version of the DOT optimizer, 7.2, is now available with GENESIS.

Topology Optimization Enhancement

No Hole Option for Filling: A new option is added for filling fabrication constraints. With no hole option, topology design will not create through holes. No hole option is available for filling from general plane (FGX, FGY, FGZ), filling from top (FTX, FTY, FTZ), and filling from bottom (FBX, FBY, FBZ).

External Eigenvalue Solver

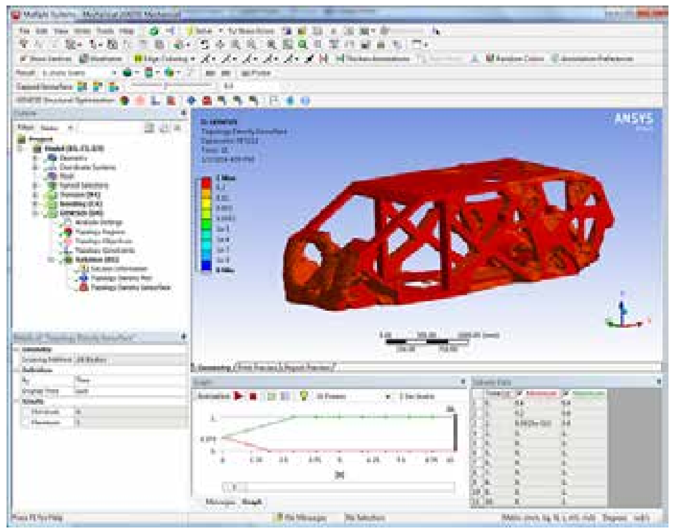
Instead of using GENESIS eigenvalue solver for frequency calculation, the user can now use ANSYS eigenvalue solver as the solution method. At each design cycle, GENESIS will call the External Eigenvalue Solver (ANSYS in this case) to solve the eigenvalue problem, while GENESIS is used for optimizing the structure.
The user can use EES to solve optimization for Modal, Harmonic, or Random Vibrations.

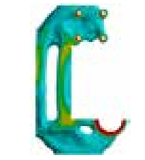
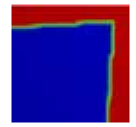
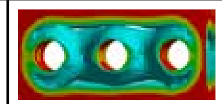
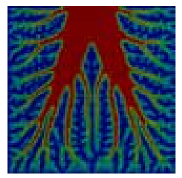
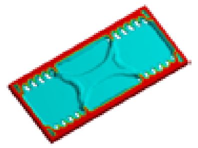
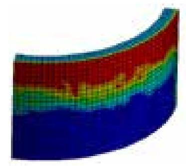
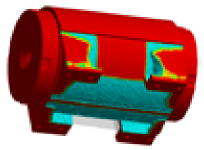
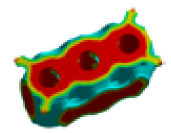
Standalone ESL for ANSYS

Instead of solving the ESL based optimization within ANSYS environment, the user can export the necessary files and run ESL for ANSYS as a standalone program. The user also needs to set environment variables pointing to the GENESIS executable location and ESL for ANSYS install location on the machine where the optimization will be performed.

Support for ANSYS RSM

In previous versions, ANSYS RSM is only supported for optimization using GENESIS solver only. In this new version, the user can run optimization based on ESL method or EES method which requires both GENESIS and ANSYS through RSM tools. The user needs to set environment variable called VRAND_ESLANSYS_INSTALL on the RSM server machine. This environment variable should point to the location of "eslansys" folder which comes with the extension installation.



File Name	Problem Title	Special Features	Figure
ATP019	Topology Optimization with Strain Constraints	Define strain constraints	
ATP020	Topology Optimization for Heat Transfer	Define Temperature as Objective	
ATP021	Topology Optimization for Thermal-Mechanical Applications	Optimize for Tem-perature and Stiffness	
ATP022	Topology Optimization with Heat Transfer Compliance	Define Heat Transfer Compliance as Objective	
ATP023	Topology Filling Fabrication Constraints: Allow through holes or No Holes	Filling Fabrication Constraints: Allow through Holes or No Holes	
ATP024	Topology Optimization with Internal Heat Generation	Thermal topology parameter for Internal Heat Generation	
EES001	Freeform Optimization with External Eigenvalue Solver	Optimization with External Eigenvalue Solver (EES)	
EES002	Topology Optimization with Prestressed Modal Analysis	Optimization with External Eigenvalue Solver (EES) for prestressed modal analysis	
AAT001	Use ANSYS Superelement: the Part/body Reduced as Superelement is Part of the ANSYS Mechanical Model	Steps to use ANSYS superelement in GENESIS	
AAT002	Use ANSYS Superelement: the Part/body Reduced as Superelement is NOT Part of the ANSYS Mechanical Model	Steps to use ANSYS superelement in GENESIS	
ETL001	Topology Optimization with Imported Thermal Load	Define ETL for using imported load from transient thermal analysis	

Support for ANSYS Superelement

Substructuring is a procedure that condenses a group of finite elements into one element represented as a matrix. This matrix is called superelement. In GSAM/GTAM, the user can use superelement generated by ANSYS in a GENESIS analysis or optimization. Typically, the user can reduce the non-designable region as superelement. With GSAM/GTAM extension, the procedures for utilizing ANSYS superelement is differentiated in two cases based on if the reduced part/body is part of the ANSYS Mechanical model.

Using Updated External Thermal Load (ETL)

In case of thermal-mechanical coupled systems, instead of using GENESIS for thermal analysis, the user can now use ANSYS thermal analysis to compute the updated "Imported Body Temperature" for static system. The optimization will be performed for the static system using GENESIS.

Summary

The key new features and enhancements include the following:
Topology Optimization for Heat Transfer Analysis: Heat transfer analysis now is supported for topology optimization. Temperature response can be used for topology optimization with heat transfer analysis. An additional new response, Heat Transfer Compliance (HTC), is added for heat transfer load case.

New Fabrication Constraints for Topology: New no-hole option is added for filling fabrication constraints. This new option will not create through holes for filling.
External Eigenvalue Solver (EES): Now it is possible to use ANSYS as external eigenvalue solver during optimization for Modal, Harmonic, or Random Vibrations.
Standalone ESL for ANSYS: Optimization based on ESL method for ANSYS now can be run directly from command line instead of within ANSYS Mechanical.
RSM Support for ESL based Optimization: ESL based optimization is supported for ANSYS Remote Solve Manager (RSM).
RSM Support for EES based Optimization: EES based optimization is supported for ANSYS Remote Solve Manager (RSM).
Support for ANSYS Superelement: ANSYS superelement is now supported in GSAM/GTAM.
Using Updated External Thermal Load (ETL): The user can now use ANSYS thermal analysis to compute the updated 'Imported Body Temperature' for optimization of thermal-mechanical coupled systems
Updated DOT Optimizer: GENESIS now uses DOT 7.2 which is the latest version of the DOT optimizer.

For more information on GENESIS and other VR&D products, please contact: Federico Bologna, EnginSoft
f.bologna@enginsoft.com

EnginSoft and Artificial Intelligence

On the 12th of April, EnginSoft took part to the conference on Artificial Intelligence organized by E4 Computer Engineering (http://www.e4company.com/Press/Events.aspx), NVIDIA and the University of Bologna, in collaboration with IBM: "INTELLIGENZA ARTIFICIALE: DALL'UNIVERSITÀ ALLE AZIENDE. LA RIVOLUZIONE DEL DEEP LEARNING" ("ARTIFICIAL INTELLIGENCE: FROM UNIVERSITY TO COMPANIES. THE DEEP LEARNING REVOLUTION").

The event was centered on the state of art of deep learning techniques (with speeches from University teachers) and industrial applications, and EnginSoft presented a work on classification and data-mining from satellite images.

In recent years, the development of new technologies has provided a considerable amount of data on environmental parameters, and high-resolution satellite images which allow not only to study land usage, but also to plan development strategies. For example, this leads to being able to develop optimal land usage plans, or track green areas evolution within a region, which in turn leads to having objective data for risk evaluation

Figure 1 - Sample image (representing Bari) used for supervised and unsupervised classification

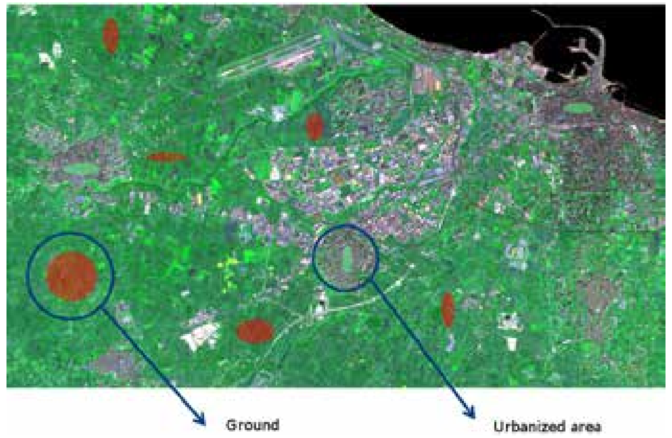


Figure 2 - Selection of training areas for supervised classification (ground and urbanized area)

and decision making.

In the presentation, EnginSoft reports on its experiences (acquired through the regional project Apulia Space) on extracting information from satellite images through data mining technologies and machine learning. I would like to point out that the techniques described herein may also effectively apply to other domains, provided the algorithms are suitably re-adapted and contextualized to the data available to the customer.

In the first part, the most important aspect of the supervision of the classification was highlighted. In this type of classification, the analyst identifies in the original image (Figure 1) homogeneous samples representative of the different types of objective soil (Figure 2). These samples are called training areas because they are the areas where the classifier is "trained" to get the final result (Figure 3). The selection of appropriate training areas is based on the analyst's familiarity with the geographic area and his knowledge of the soil cover types present in the image. This way, the analyst controls the categorization of a set of specific classes. The numerical information contained in all the spectral bands corresponding to the pixels that make up these scopes are used to "train" the computer to recognize spectral areas similar to each class.

In the unsupervised classification, the classification supervision process is reversed. Spectral classes are first grouped only on the numerical information in the data and then merged from the analyst to the information classes. Normally, the analyst specifies how many groups or clusters are to be searched for in the data. In addition to specifying the desired number of classes, the analyst can also specify the parameters for separation distance between clusters and variation within each cluster. The final result of this process applied to Figure 1, in which 4 clusters were chosen, is shown in Figure 4.

The many examples shown at the workshop were well-diversified, giving good examples of how similar techniques can be applied to different industrial sectors (finance, business asset maintenance, biomedical, ...). All these speeches were linked to the common subject of artificial intelligence and the role it plays. These methods are at the core of Deep Learning and Industry 4.0 technologies.

For more information: Manolo Venturin, EnginSoft
m.venturin@enginsoft.com

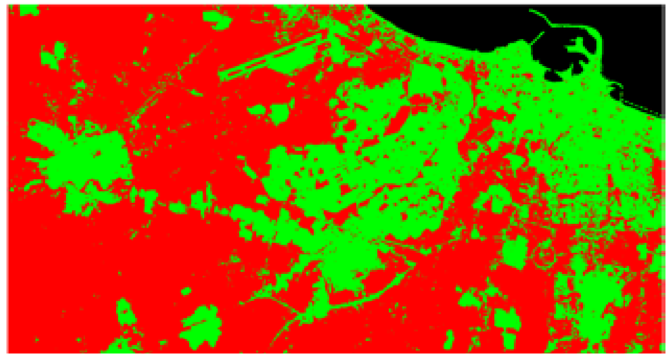


Figure 3 - Results of the supervised classification (segmentation mask)

Figure 4 - Results of the unsupervised classification with 4 classes

2017 INTERNATIONAL CAE CONFERENCE AND EXHIBITION

Beyond Engineering Simulation

One can easily argue that an event which runs into its 33rd year, addressing a community of experts is true to its role of mirroring the way Simulation Based Engineering and Science (SBES) has changed the world of engineering design. An event which has evolved from its initial scope of covering industrial CAE has transformed beyond this, exploring the widespread use of advanced technologies and Engineering application in all sectors, whose combined force is causing a true revolution in modern engineering practices and excelling scientific fields.

Indeed, simulation performs a central role in all aspects of the product lifecycle, from concept, through to engineering design and into its operations. Saving time and money by performing simulations earlier in the design cycle is not the only advantage from the technologies available today: simulation lets you delve further, to create a "digital twin" of your product, providing insight into performances and valuable knowledge in respects to maintenance. Progressively the physical world and the digital world merges as it is pushed by new technology areas such as the Internet of Things, additive manufacturing, big data, and Industry 4.0.

Such reflection of the engineering landscape drives the format of the event. The varied proposal of high-quality cultural content: from the plenary session and the industry-related sessions is populated by inspired speakers -providing leading insights, success stories showcasing engineering applications and/or technology adopted to embrace and overcome industry challenges. For an exhibition – which has proven to be exceptionally popular, will be almost double the size this year. In addition, in terms of the Research Agorà – where representatives of project Consortiums in the field of smart manufacturing, biomedical, high performance computing, energy and big data and analytics share their project concepts and achievements, complemented by the Poster Award. An award open to all students, graduates, researchers and/or faculty members from all Academic Institutes and Research Centres to help disseminate and engage the adoption of Engineering Simulation –will be accepting sponsors from industry for the first time, bridging the gap between the industrial and academic world. Finally, the Best Practice area – where methods, design approaches and techniques will be discussed in an interactive framework, designed to show how to get superior outcomes from the simulation and to comply with industry standards, opening the window on the subject of Industry 4.0. Furthermore, a dedicated area, including a session and exhibition on additive manufacturing, including short courses on topical issues like fatigue for composite materials or geometric dimensioning and tolerancing to the opening towards the EC trends and offering.

All this were facts in the former editions and offers a glimpse into what we will expect in the next edition.

But, beyond these facts, the difference is the enthusiasm that goes through all in initiatives, an atmosphere that can only be captured by taking part. This is a truly unique experience, and often passionately expressed by participants and supporters of the event.

Don't come to just join an event, join a community that always excited about the future and will continue to encourage knowledge transfer!

For more information:
www.caeconference.com