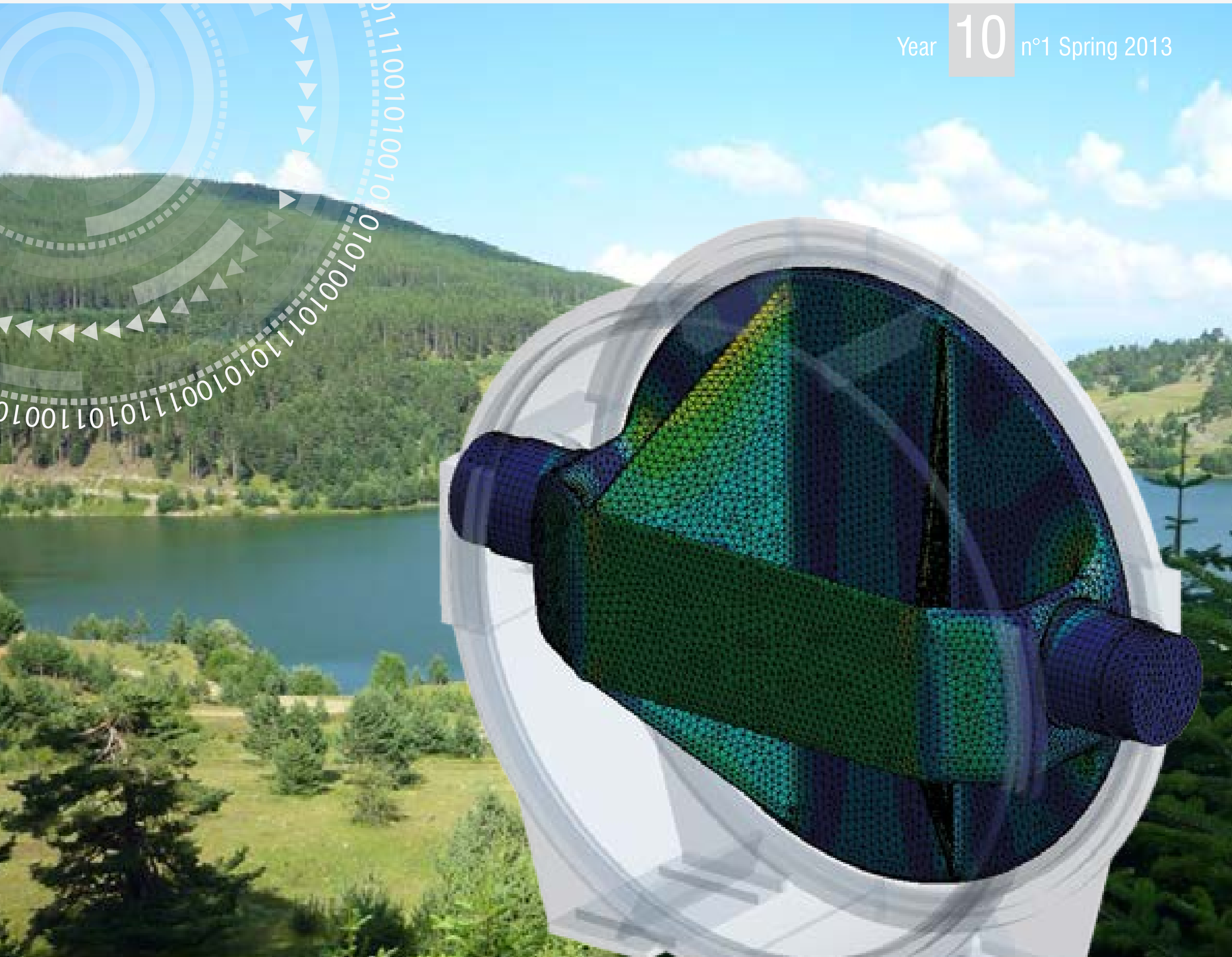
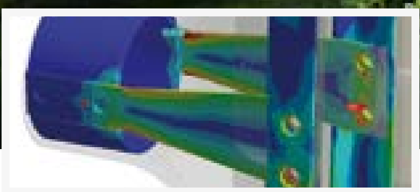
Newsletter

Simulation Based Engineering & Sciences

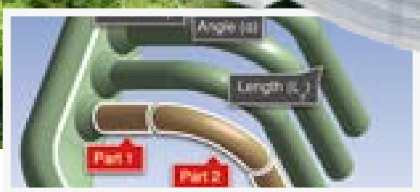
Year 10 n°1 Spring 2013





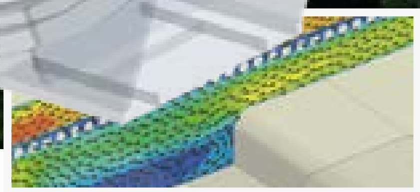
Vin Service and EnginSoft Partners for Innovation in **Beverage Dispensing Equipments**

Second **Hydro Power Plant** in Turkey is taking off: Design of valves and actuators



Air Intake Manifold: Design and Optimization with a Powertrain Platform

Mechanical Optimization of the injection system in a **compression molding machine**



Burners and Combustion Systems Major efficiency and Reduced Management Costs with Simulation

A Multi-Physics Approach for **Reducing Fuel Consumption** in Aero Engines

FLASH

The year 2013 not only promises new exciting projects and initiatives for the engineering simulation community, it also marks the 10th anniversary of the EnginSoft Newsletter.
We look back on 10 years of research work and networking to collect a variety of relevant topics for our readers. Our primary goal has always been and is to present a wide range of engineering expertise, technologies and latest advancements from our fields of business, research and science.
In the eyes of EnginSoft and the Editorial Team, engineers and our readers are the creators that inspire our work. In this spirit, I encourage you to please talk to us whenever you wish to discuss new developments, activities or publications for the Newsletter!
This Edition brings us news on the second hydro power plant in Turkey, how it is being realized with the support of experts from across Europe - EnginSoft is proud to be one of them.

We hear about the mechanical optimization of an injection system, a multiphysics approach for reducing fuel consumption of aero engines at Avio, and how Bombardier achieves the best design with modeFRONTIER. IK4-Tekniker, Abamotor Energía and CMT-Motores Térmicos from Spain present their application to model and optimize a two stroke engine, while Pierburg Pump Technology Italy explain their dynamic CFD analysis and Vin Service started a partnership with EnginSoft for innovation in beverage dispensing equipments. We hear about Magnetti Marelli's design and optimization work for an air intake manifold, and how SITI - B&T Group apply simulation to achieve higher efficiency of their burners and furnaces. Furthermore, EnginSoft Italy outlines the use of engineering analysis in a casting process.
This issue also features an interview with Carpenteria Industriale Bresciana, manufacturers of pressure vessels, a success story of EnginSoft UK, the European Project Vocal Fan, and our new partnership with Nafems. For an overview on the latest technologies and developments in our fields, please refer to the ANSYS software news, the articles about Mechatronics, the supercomputer of Stanford's Engineering Center for Turbulence Research, Charles, Scilab, Mercury-DPM and Cetol6.
The sharing of knowledge with our customers and partners is an essential part of our work. This is why EnginSoft participates in technology events throughout the year and organizes training courses, seminars and workshops; many of them are listed for you in this Newsletter.

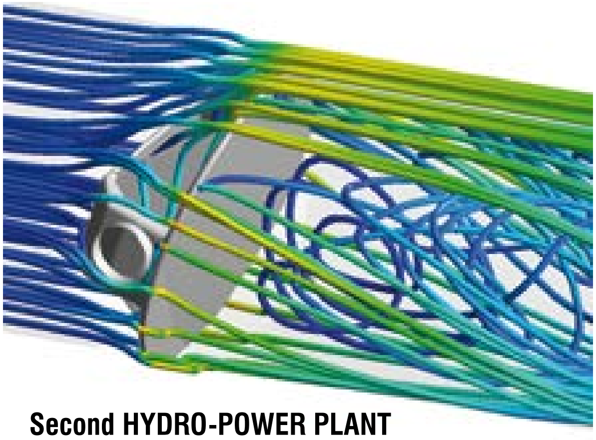
The ANSYS User Group Meeting Italia 2013 will take place on 20th & 21st June at Salsomaggiore Terme (Parma). The Meeting is targeted at ANSYS users who wish to learn about the latest developments of the technologies. Please visit ANSYS Italy and EnginSoft at the main central booth in the exhibition.

Finally, we are delighted to announce our major annual get-together: the International CAE Conference 2013. Please mark the dates, 21st – 22nd October, in your diary. You will receive your personal invitation from us soon!

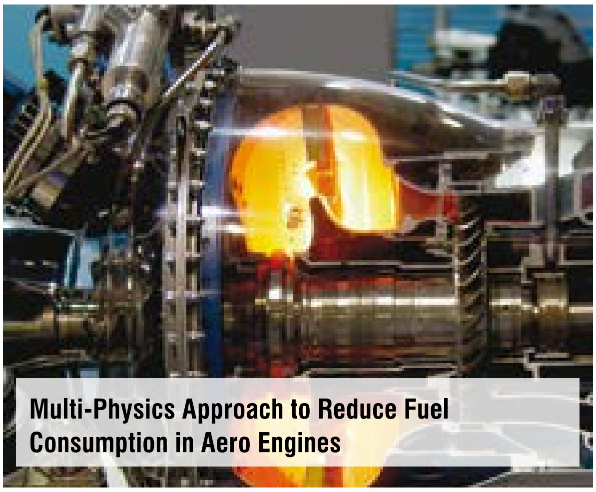
Stefano Odorizzi
Editor in chief

Sommario - Contents

CASE HISTORIES



Second HYDRO-POWER PLANT Turkey is taking off



Multi-Physics Approach to Reduce Fuel Consumption in Aero Engines

IN-DEPHT STUDIES

SOFTWARE UPDATE

RESEARCH AND TECHNOLOGY TRANSFER

PARTNER NEWS

TRAINING

EVENTS

Newsletter EnginSoft
Year 10 n°1 - Spring 2013

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

Advertisement

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

EnginSoft S.p.A.

24126 BERGAMO c/o Parco Scientifico Tecnologico
Kilometro Rosso - Edificio A1, Via Stezzano 87
Tel. +39 035 368711 • Fax +39 0461 979215
50127 FIRENZE Via Panciatichi, 40
Tel. +39 055 4376113 • Fax +39 0461 979216
35129 PADOVA Via Giambellino, 7
Tel. +39 049 7705311 • Fax +39 0461 979217
72023 MESAGNE (BRINDISI) Via A. Murri, 2 - Z.I.
Tel. +39 0831 730194 • Fax +39 0461 979224
38123 TRENTO fraz. Mattarello - Via della Stazione, 27
Tel. +39 0461 915391 • Fax +39 0461 979201
10133 TORINO Corso Moncalieri, 223
Tel. +39 011 3473987 • Fax +39 011 3473987

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

COMPANY INTERESTS

CONSORZIO TCN
www.consorziotcn.it • www.improve.it

EnginSoft GmbH - Germany
EnginSoft UK - United Kingdom
EnginSoft France - France
EnginSoft Nordic - Sweden
www.enginsoft.com

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

RESPONSIBLE DIRECTOR

Stefano Odorizzi - newsletter@enginsoft.it

PRINTING

Grafiche Dal Piaz - Trento

The EnginSoft NEWSLETTER is a quarterly magazine published by EnginSoft SpA

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

Contents

Second HYDRO POWER PLANT Turkey is taking off

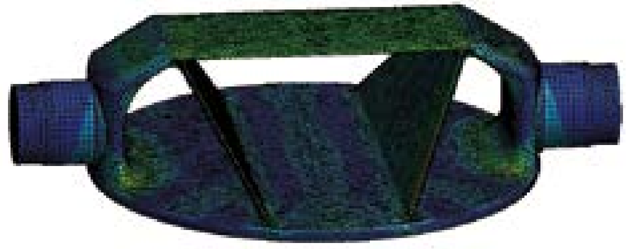
High quality engineering speaks Italian

The Kargi Project that consists of the construction of the second major hydroelectric power station in Turkey, will be completed in the second half of 2013 with the direct involvement of expertise, resources and technologies coming from all over Europe.
The power station located about 200 km northeast of Ankara, will take advantage of a geodetic head, partially natural and partially artificial, of about 80m provided by the Kizilirmak Basin. It will allow two Francis turbines with vertical axis and their respective alternator groups, to generate over 110MW of electric power and an annual average productive capacity of about 470GWH.
EnginSoft, with its Oil&Gas and Civil Engineering Competence Center, has supported Carpenteria Industriale Bresciana Srl (CIB) and the Austrian main Contractor Voith Hydro GmbH, in the design and following production of the valves and related actuators, which are necessary to control the water outflow coming from the dam located 11,5 km away.
The pressure pipeline has been totally placed inside a tunnel, excavated by means of a huge drill, almost 150m long and able to obtain a 10m diameter in just one pass.
The valves, characterized by a fluid flow section of 4100mm flow and a weight of 75 tons, have been manufactured in a record time by CIB Carpenteria Industriale Bresciana in electro-welded steel. They will be installed at the end of the pressure pipe in order to disconnect, if necessary, the entry flow of the turbine which, under heavy conditions, might reach the considerable flow of 84.000 l/s.
Considering that the plant is located in Turkey, in an area with a high risk of earthquakes, and that even under limited-use conditions, it has to guarantee safety and an average life of some decades, both design and sizing have to be accurately analyzed, nothing could be left to chance. Even the "butterfly" shape (of the disk which laminates the water flow) has been designed to minimize load losses/pressure drops when the valve is open, and, at the same time, to guarantee the stiffness and the water tightness when the valve is closed. These characteristics are totally in disagreement! The structural sizing has been completed with fluid-dynamic analyses.
These have allowed the designers to develop the butterfly-shaped cut-off profiles according to the hydrodynamic characteristics that are adequate to reduce the load loss coefficient to a minimum extent. The profile morphology, particularly enhanced in order to meet the project specifications, required targeted interventions to achieve shape optimization and, as a consequence, to obtain a real hydro-dynamic and highly efficient profile out of a rough manufactured component.

The fluid-dynamic behavior has been simulated for different partitions of the cut-off. Two of them proved to be particularly relevant and worth of an in-depth description:

- Maximum opening, with cut-off walls parallel to fluid motion. In other words, the normal functioning condition in which the maximum allowable coefficient of valve loss has been studied and detected.
- Intermediate position, here the maximum mechanical stress is transmitted to the "butterfly" actuator. This angular position ranges from 20 to 25 degrees with respect to the total closure position.

The team headed by Livio Furlan, on Voith's request, has investigated and implemented a hydraulic seal system on the cut-off hubs that allows seals replacement when the plant is active. This brilliant design will allow the Turkish technicians to maintain the plant without interrupting production. The first inspection of the manufactured products has been performed by Voith Hydro at the builder plant in Brescia. The valves have been statically tested at a test pressure of 1.5 times of the design on-stream pressure by means of test equipment that was set up ad hoc for this purpose. Practically speaking, the test case consisted of a huge bell, flanged to the valve, in which water was injected and artificially led to a test pressure through pumps and load

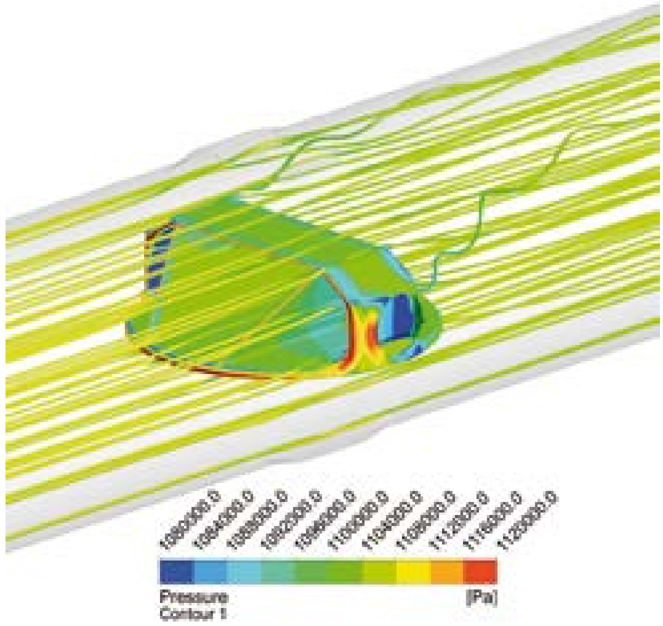


Von Mises stress distribution at Valve Disc for Valve in closed condition

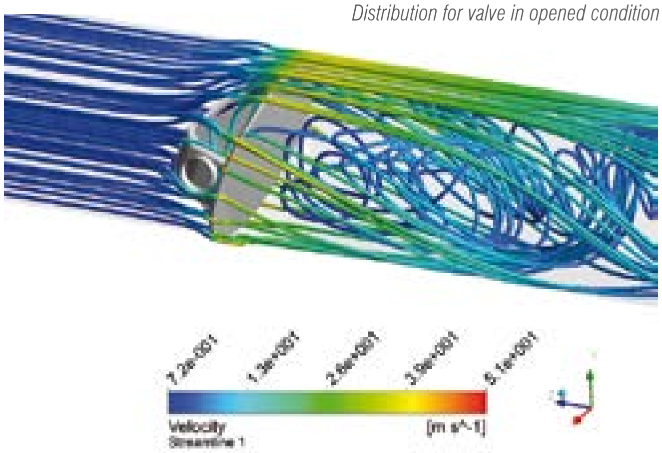
control systems. In addition to the checks for tightness, the inspectors from Voith have also experimentally detected the correlation between the experimental and the theoretical distortions. This has been done using simulation and through constantly monitoring the measurement systems throughout the test.
So what can we say? …That high quality engineering speaks Italian, also in Turkey.
The team of engineers that have developed the Kargi Project will be present at the EnginSoft booth at the OMC 2013 – Oil&Gas, Offshore and Inshore Exhibition, scheduled March 20 – 22 in Ravenna, Italy. We look forward to meeting our customers and friends who want to hear more about this and other completed projects!

For more information:
Livio Furlan - EnginSoft
info@enginsoft.it

Livio Furlan, Giulio Morandina, Emanuele Bassi from Oil&Gas EnginSoft Competence Centre



Distribution for valve in opened condition



Francis Turbines

Francis Turbines are used for medium-low heads, between roughly 5m and 120m, when flows range from 100l/s to over 10.000l/s. The Francis turbine, as well as the Kaplan turbine, is a reaction turbine in which water pressure varies from rotor inlet to outlet. Through adduction in a spiral shape, water is distributed on the rotor circumference in a uniform way, and then channeled to the inside of the rotor, through the distributary. At the same time, the runner blades of the distributary allow the whirled water quantity to be regulated. Inside the rotor, water is diverted from the blades causing an energy transfer to the rotor of the energy contained in the water. The water gets out of the rotor in axial direction through the ejection pipe, taking advantage of the relief between the machine axis and the water lower surface. The so-called aspiration head presents itself an under-pressure at the rotor outlet. Considering that the rotor blades are fixed and cannot be adjusted according to the flow angle, which varies depending on the distributary blades position, the efficiency field of the Francis turbine is quite limited.
As a compensation, Francis turbines have the best efficiency peak of all turbines, which when used for huge plants, can reach a range of 94% and above.

Carpenteria Industriale Bresciana Un'eccellenza tutta italiana

Localizzata a Castegnato, in provincia di Brescia, la CIB – Carpenteria Industriale Bresciana – è un'azienda specializzata nello sviluppo e manifattura di grandi componenti destinati al completamento di centrali elettriche, impianti minerari e siderurgici. La produzione consiste nella progettazione e costruzione di grossi corpi saldati finiti di lavorazione meccanica.
Dalla sua fondazione, avvenuta nel 1968, l'azienda ha orientato la propria operatività dalla carpenteria per l'edilizia, prevalentemente strutture in acciaio per capannoni ed insediamenti industriali, a produzioni via via più specializzate come le condotte forzate per l'idroelettrico, caldareria, serbatoi in pressione ed altro.
In parallelo, anche il parco clienti di CIB si è trasformato. Oggi l'azienda vanta collaborazioni, dirette, con multinazionali dell'energia e dell'impiantistica in generale. Tra i principali: Alstom, Andritz Hidro, Outotec, Thyssen, SMS Siemag, Voith e molti altri.
Per EnginSoft Newsletter abbiamo intervistato il geom. Luciano Mattanza, consigliere delegato dell'azienda e l'ing. Rossana Mattanza project manager della Carpenteria Industriale Bresciana.


Fig. 1 - L'Ing. Rossana Mattanza e il Geom. Luciano Mattanza della Carpenteria Industriale Bresciana

Ing. Mattanza cosa differenzia la Catena del Valore espressa sul mercato da CIB rispetto la concorrenza?
Il primo fattore differenziante è la gran passione per ciò che facciamo. Inoltre la nostra famiglia, che governa CIB sin dalla sua fondazione, considera il sistema-azienda come un unico grande organismo nel quale clienti e risorse umane impiegate, oggi oltre 100, hanno pari dignità e sono considerati elementi fondamentali per il successo. Riteniamo strategiche anche le relazioni instaurate con l'indotto la cui fiducia e reciproca stima si sono consolidate nel corso degli anni. Fortunatamente - ed è un modo di dire perché chi fa impresa ripone poca fiducia alla fortuna… - l'azienda è solida e l'attuale portafoglio ordini ci consente di guardare al futuro con serenità. I nostri clienti, internazionali soprattutto, credono in noi perché azienda in grado di garantire il rispetto dei tempi di consegna e al contempo elevati standard qualitativi. Capacità che la concorrenza, soprattutto di paesi emergenti, non è ancora in grado di esprimere. La competitività economica e finanziaria è quindi una conseguenza derivata.

In questo momento di crisi tutti parlano d'innovazione come la panacea a tutti i mali. Ci esprime la sua personale definizione?
Noi innoviamo continuamente e lo facciamo da anni! Questo, ritengo sia un altro ingrediente fondamentale per il nostro successo. Siamo in procinto di installare nuove macchine (tra queste un tornio verticale in grado di lavorare particolari da 16 metri di diametro!) e raddoppiare la superficie produttiva coperta. Questi investimenti, a 6 "zeri", aumenteranno le nostre capacità in linea con bisogni e necessità di mercato. Siamo, a tutti gli effetti, un'azienda in contro tendenza. Anche nella carpenteria metallica dell'acciaio - una branca della meccanica che molti a torto considerano povera - le evoluzioni tecnologiche sono continue. Ad esempio sperimentiamo, continuamente, nuovi processi di saldatura e materiali d'apporto. Ma non solo. Grande attenzione ed investimenti sono anche orientati verso strumenti, tecnologie e metodologie di verifica e controllo qualità. Tra questi i controlli non distruttivi: radiografici, ultrasuoni, magnetici, liquidi penetranti, ecc.

Qual è la sua personale opinione in merito al contributo erogato dagli strumenti di simulazione computerizzata, in generale, nel perseguire l'innovazione di prodotto e/o processo?
Oggigiorno le attività di progettazione eseguite da risorse interne sono inferiori che nel recente passato. Abbiamo compreso che per alcuni progetti, o parte di questi, è preferibile avvalersi di efficaci specialisti quale l'EnginSoft. Siamo ovviamente dotati di sistemi di progettazione e simulazione computerizzati CAD, CAM e CAE con i quali, oltre ad alimentare le necessità interne, verifichiamo quanto commissionato all'esterno. La nostra dotazione di soluzioni d'avanguardia, ci consente di dialogare efficacemente con il parco-clienti e ci aiuta nell'analisi dei probabili scenari di progetto ovvero nella loro preventivazione economica e temporale.

Può darci un accenno ai vantaggi, globali, conseguiti da CIB attraverso l'impiego del CAE (o della simulazione in generale) in termini di competitività generale e/o "time–to-market"?
Questi strumenti e la loro padronanza sono fondamentali per competere e rassicuranti per la clientela. Non riceviamo più, da tempo memorabile, commesse nelle quali non sia richiesta la simulazione CAE e verifica di correlazione tra dati sperimentali e teorici. Questo know-how è oggi determinante per competere. È molto più importante offrire garanzie di qualità duratura nel tempo che il suo costo di produzione nell'immediato. Gli impianti e i beni che sviluppiamo e costruiamo, consideri un impianto minerario o una centrale idroelettrica, devono assicurare produttività negli anni minimizzando se non annullando gli imprevisti che possano interromperne l'esercizio. La simulazione offre buone garanzia di performance in questo senso.

Un'ultima domanda: ha un auspicio particolare per il 2013?
Non posso non aver auspici: non farei l'imprenditore. La mia maggior speranza è di stabilizzazione dell'attuale situazione economico-finanziaria e della ingombrante burocrazia che non ci facilita il far impresa in Italia.

A cura della Redazione

Mechanical optimization of the injection system in a COMPRESSION MOLDING MACHINE

The machine under study is a Continuous Compression Moulding (CCM) hydraulic press for plastic cap manufacturing with extrusion of dry-blend granulated compounds. The machine performs the metering, the pelleting and the insertion in the cavities of the molded resin in order to obtain the final shape which is a semi-finished cap.
The CCM machine is made of the following components: controller, extrusion, electric box, cooling box, molding module. The machinery has two carousels: one for completing the molding of the resin and one for executing the injection of the cavity. The latter performs the portioning of the melted pellets, the transportation and the compression inside the mold. In the injection phase, a pneumatic piston will insert the melted pellet into the mold.
The pneumatic piston is the component which we will investigate in an optimization approach that we present in this article.
A critical aspect of the problem is that the compression phase to complete the piston movement only lasts fractions of a second, two milliseconds for ascending and two milliseconds for descending - this allows a maximum production capacity of 600 Pieces per Minute.
The main consequences are high speed results and huge accelerations which can lead to potentially harmful collisions.

Fig. 1 - a) Original 12 mm Piston. b) Broken Piston

I wasn't an experienced modeFRONTIER user, so I found the Optimization Wizard very useful: it supported the choice of the number of designs, of the algorithm and of the number of iterations according to the time available for the whole analysis.

Andrea Minardi
CAE specialist at SACMI Packaging Division

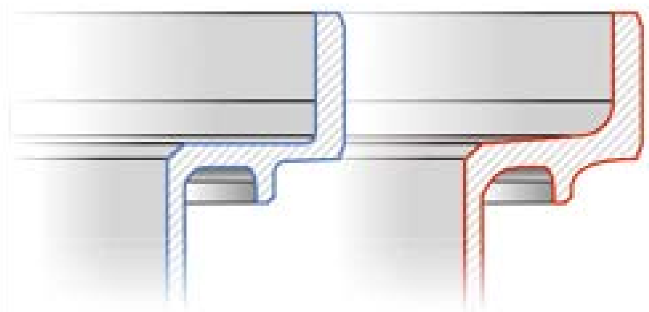


Fig 2 - Tapered Geometry (RED)

The first optimization procedure was completed by manually changing the shape of the piston in the configuration with a 12 mm diameter, in order to prevent occurring fractures as shown in Figure 1.

The design activity was carried out using Solidworks (from Dassault Systèmes) as geometry modeler and ANSYS as Finite Element Method tool.
The manual optimizations followed the mechanisms of intake and exhaust valves inside endothermic engines, where the right shape configuration of tapered surfaces is essential for improving performances. In this way, the piston region where fractures generally occur, was modeled through a tapered geometry (see Figure 2).
The solution for the design problem required a trade-off between the reduction of weight and an increased safety factor, both could be achieved with a stress analysis. The increase of the weight of a high-speed moving component led to damages in some surrounding areas. For this reason,

Fig. 3 - a) Original geometry b) Manual optimized geometry

Geometry	Minimum Safety Factor
ORIGINAL (before)	0.9
MANUAL OPTIMIZED (after)	1.2 (+33.3 %)

Table 1 - Comparison Before and After the manual optimization

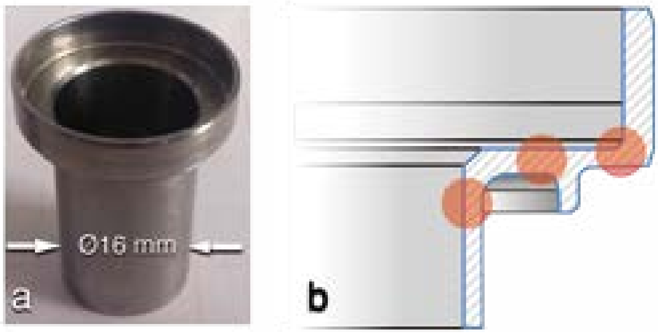


Fig. 4 - a) 16 mm piston with original geometry b) Critical regions (RED)

we decided to design a light yet strong piston. After 20 working days, the manual optimization had to be terminated due to difficulties we experienced in achieving further improvements in the performances of the part. The results are summarized in Table 1 and in the Before and After Geometry comparisons (Figure 3).

In the next phase of the study, the new component to be optimized was a 16 mm diameter piston that was initially sketched with the same original geometrical configuration (not tapered) of the 12 mm piston (before the manual shape optimization). In this configuration, the piston features some potentially harmful regions for the fatigue analysis as illustrated in Figure 4.

In addition to the geometry modeler (Solidworks) and the FEM tool (ANSYS), an optimization tool has been adopted (modeFRONTIER). The idea is to use modeFRONTIER for schematizing the design process and for defining an optimization strategy.

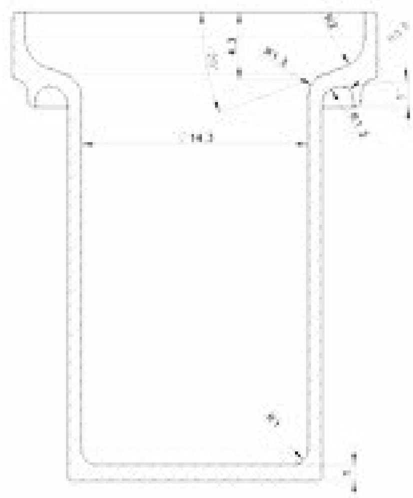


Fig. 5 - Free geometrical free parameters

The first step was to prepare a parametric geometry of the piston in order to transform the most relevant model entities into design variables that can be managed by the optimization Environment. Figure 5 shows the two-dimensional draft sketches and the nine parameters that control all the possible piston shapes. The problem is solved in two dimensions because its axial-symmetric characteristics also allow a consistent reduction of simulation time.

The second step of the automatic optimization set-up was to create a FEM model starting from the parametric geometry. Here, Workbench from ANSYS has been used as it allows the designer to lay-out the simulation process. In our present case, two simulations had to be performed (see the scheme in Figure 6): the first should solve the stress generated by the collision of the piston (on the left), while the second (on the right)

Fig. 6 - ANSYS Workbench simulation layout

represents the final run of the analysis that will supply the simulation results to modeFRONTIER.

The final step of the optimization procedure is to define the objectives and constraints along with the strategy to be used for the search of the optimum. The modeFRONTIER workflow shown in Figure 7 reflects the entire process in a single diagram: input variables (on the top), process integration and optimization strategy (in the middle), output variable, objectives and constraints (at the bottom).

With regard to the adopted strategy, an initial set of configurations was needed for initializing the optimization algorithm. The wizard of the optimization tool provided the proper choice of methods in order to pursue the objectives in a time frame of 24 hours.

The system selected the SOBOL Design of Experiments of 50 starting points (designs). Then a Multi-Objective Genetic Algorithm was used for achieving the two design objectives, the Maximum Principle Stress (X axis, see Figure 8) and the Minimal Principal Stress (Y axis, see Figure 8). The constraints were provided by the acceptable weight tolerances (+ - 10%) towards the original design. The optimization successfully completed 800 different piston configurations (which are graphically represented by a bubble in the scatter chart in Figure 8, the color scale is associated with the weight of the piston (from blue to red).

The scatter chart also presents the Pareto Frontier which contains all the optimal designs (Pareto designs). It is possible to operate the trade-off between the piston configurations within the designs. This is a subjective

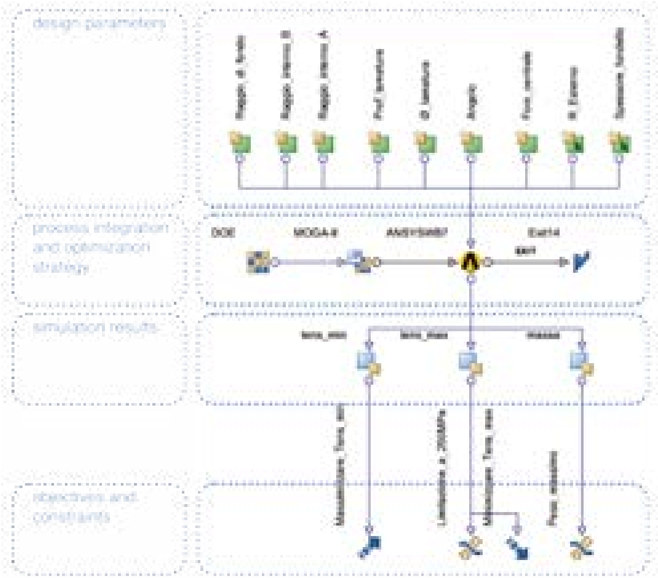


Fig. 7 - modeFRONTIER design optimization workflow diagram

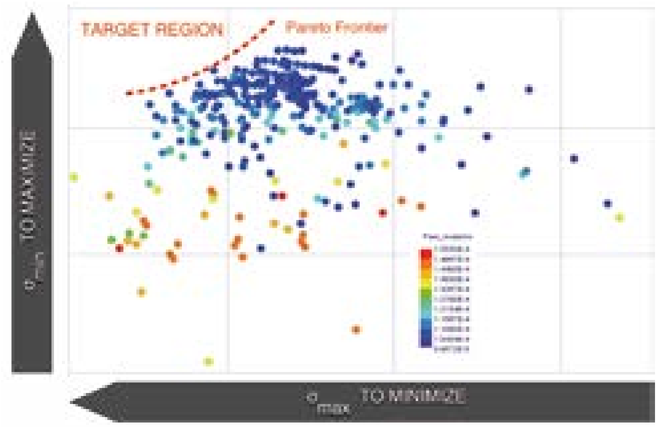


Fig. 8 - Scatter chart displaying the 800 results of the automatic optimization process

Fig. 9 - Original 16 mm piston (LEFT) and Optimal piston as result of automatic optimization process

Geometry	Minimum Safety Factor
ORIGINAL (before)	0.9
MANUAL OPTIMIZED (after)	1.3 (+44.4 %)

Table 2 - Comparison Before and After the automatic optimization on the 16 mm piston

Fig. 10 - Correlation Plot

choice that has been carried out in order to select the final design obtained through an automatic optimization procedure (see Table 2). The optimal piston selected for the comparison with the original one is shown in Figure 9.

With modeFRONTIER it is also possible to obtain statistical analysis results across the design variables. The statistical plot in Figure 10 shows the direct and inverse relationship between the problem variables. More specifically, it features in colors (from red to blue) and numbers (from 1 to -1) the correlation between the input parameters and the output measure. For example, by increasing the internal diameter of the piston it leads to a consistent (0.77 means high direct relationship) increase of the Minimal Principal Stress and an even greater (-0.99) reduction of the total mass of the piston.

Conclusion

In conclusion, we can say that we have obtained a sufficient improvement on the 16 mm piston while substantially saving time for completing the tasks. Moreover, by adopting a design framework as the one provided by modeFRONTIER, the designer can focus more on the pre and post-processing of the optimization results. It is no longer necessary to change, again and again by hand, each parameter at a time for increasing the safety factor of the component under study. This leads to a better exploitation of the design cycle time (as illustrated in Table 3) and to an optimal solution, both through objective optimization and subjective selection of the best compromise among the available results.

ing. Andrea Minardi - SACMI Group
ing. Francesco Franchini - EnginSoft

For more information:
Francesco Franchini - EnginSoft
info@enginsoft.it

	MANUAL OPTIMIZATION	OPTIMIZATION WITH modeFRONTIER	OVERALL OUTCOMES
VIRTUAL ANALYSES TESTED	about 30	800	**+2667%** better exploitation of simulation resources
PERFORMANCES IMPROVEMENT	+33.3 %	+44.4 %	**+33%** better results
TOTAL TIME TAKEN TO COMPLETE THE TASK	20 days	4 days	**500%** better exploitation of design time

Table 3: Comparison between manual and automatic optimization.

Vin Service and EnginSoft Partners for Innovation in Beverage Dispensing Equipments

Founded in 1976, Vin Service is a leading company in the field of beverage dispensing equipment. It offers a broad range of beer equipment with a wide variety of designs and styles, post-mix dispensing equipment and a complete line of accessories for complete installations along with a professional design and development service from project concept to implementation. The company takes care of the design, development and engineering, manufacturing, logistics, quality control, marketing and sales. The entire production is realized inside the company plants, which cover an area of 8500mq.

Fig. 1 - Dispensing beer fount

EnginSoft cooperated with Vin Service in the project of an innovative dispensing beer fount. The customer was a global corporation leader of the beer industry.

Most innovative aspect of the project was introducing high performance polymers for structural applications, into a market which for decades was based on heavy brass casting (very often oversized) or complex and expensive steel frames. EnginSoft performed FEM structural analyses on different configurations of the design. Analyses were realized by mean of a complete 3D FE model, considering different load cases (preload on condensation tray, vertical and horizontal loads at the top of fount), different geometries and materials. Model nonlinearities (geometric nonlinearities, nonlinear contacts between parts) were also considered.

Fig. 2 - FE model of the fount

Assessments were done both on maximum stresses reached on various parts of the fount, and on maximum displacements reached for different load cases. The limits were imposed by stringent customer specifications.

EnginSoft experts also cooperated with Vin Service in the choice of most suitable polymeric materials for this kind of application. For instance, initial structural steel frame proposed by final customer was replaced by a glass fiber filled polyamide component. The analyses were completed by provisions of behavior of plastic structure in aging conditions (forecast to 5 years) and in fatigue conditions, since the

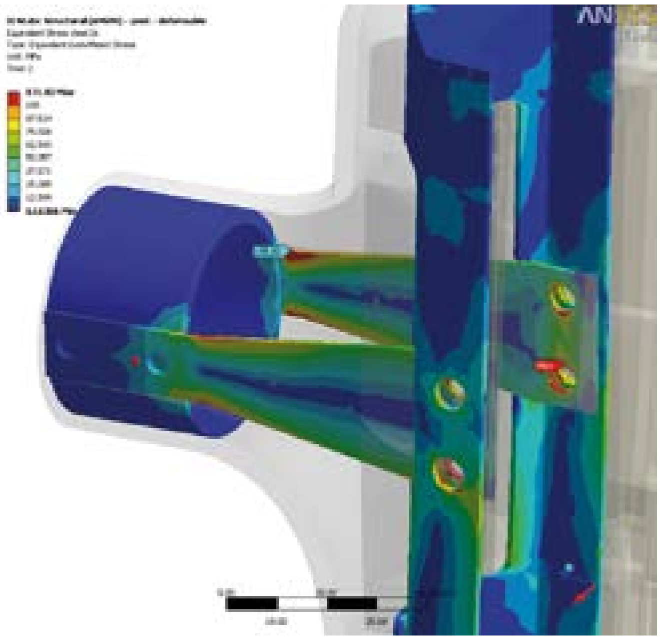


Fig. 3 - Stress in original steel frame

component has to withstand a great number of load cycles. These verifications allowed to avoid any remaining doubt about the reliability of final design. The CAE application in this project allowed to reach the customer imposed targets. Vin Service won the contract convincing a global company to trust in an Italian SME. More, Vin Service reduced costs and technical risks, introducing and validating the method of Computer Aided Engineering in a SME active in the niche market of beverage dispensing systems. Good achieved results allowed Vin Service and EnginSoft to continue their cooperation in other innovative projects in the same industry sector.

For more information:
Maurizio Facchinetti - EnginSoft
info@enginsoft.it

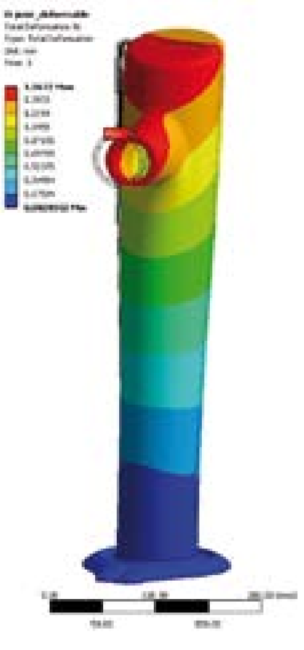


Fig. 4 - Displacements result – example

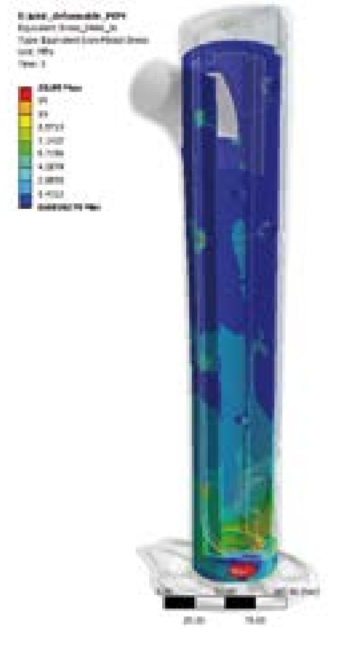


Fig. 5 - Stress in final polymeric structural component

EnginSoft in the food&beverage industry

Besides previous case, mainly focused on structural problems in beverage dispensing, EnginSoft works with the world's leading players in the food&beverage industry.
For these companies, food safety, process robustness and productivity are the most important factors of their research and development. Food safety for the consumer is the first priority. Bottles, packaging, dispensers, machines have to be washed and in some cases also sterilized. Moreover, aseptic or non-oxidant conditions have to be assured for perishable food. All these issues can be addressed and food safety can be guaranteed with the support of simulation. Computational fluid dynamics can give an insight in all these processes, to study for example the interaction of chemical species with packaging, machines and food. Thermal management is another topic that is relevant for many applications, it involves several disciplines from electromagnetism to structural and thermo fluid dynamics.
Also, process robustness is a key factor which leading companies in these sectors that use industrial lines to fill bottles and packaging with their products, have to guarantee to their customers. Each component, the whole line and the process itself, have to be reliable and robust.
Last but not least, productivity has to reach the limit. The production rate can reach 20.000 packages/bottles per hour depending on the type of food. Hence, every step has to be optimized in terms of efficiency, which means reducing time and keeping the process quality at the maximum level. The ANSYS technologies and EnginSoft's experiences in engineering simulation can build synergy, to cover multidisciplinary applications and to give simulation the right role, which is to support and drive the design process in advance with respect to physical prototyping thus reducing development time and costs.

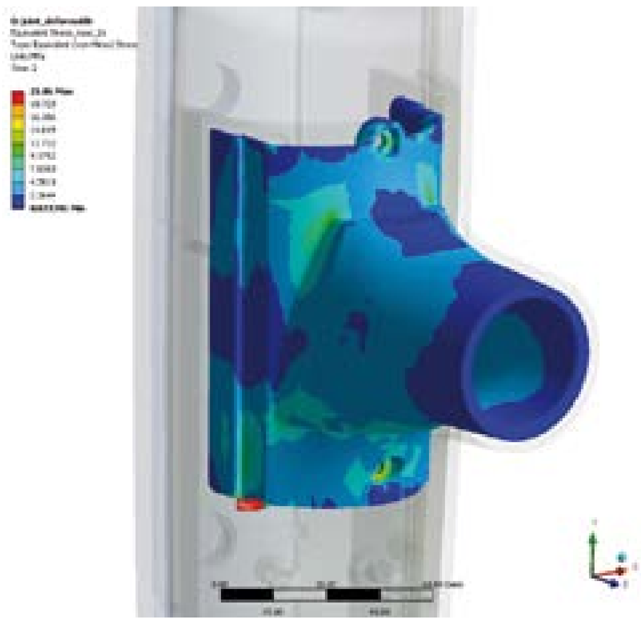


Fig. 6 - Stress result in a fount component

A Multi-Physics Approach to Reduce Fuel Consumption in Aero Engines

The recent evolutions of aeronautic engines require increased efficiency in order to reduce fuel consumption and to meet the targets of future emission levels. One of the most important components affecting the efficiency of jet engines is the gas turbine. A high turbine efficiency can be achieved by means of a performing blade designs as well as by a proper management of mass flows during the entire engine flight. Regarding the latter factor, a proper mass flow management is achieved by controlling the clearances in turbine modules, i.e. by controlling the distances between static and rotating parts of the turbine. In particular, a high turbine efficiency can be achieved by keeping the clearances at the minimum allowable value in any flight conditions. Clearances can assume different heights during the flight mission depending on the deformation and displacement of the various turbine components. The active control of clearances heights can be performed by controlling the temperature of the static components through an effective cooling system. The temperature distribution in the turbine, indeed, determines deformations in the solid structure which modifies the clearances and, as a consequence, the flow field. At the same time, the flow field influences the temperature distribution in each turbine component. Moreover, considering that typical diameters of aeronautic turbines are more than 1 m while typical clearances heights are less the 0.5 mm, it is clear that high design accuracy and absolute reliability are required. For all these reasons, only a numerical multi-physics approach capable to model the entire system early in the design phase and capable to account for fluid-dynamic, thermal and deformational behavior at the same time can guarantee the required clearance design accuracy and reliability.

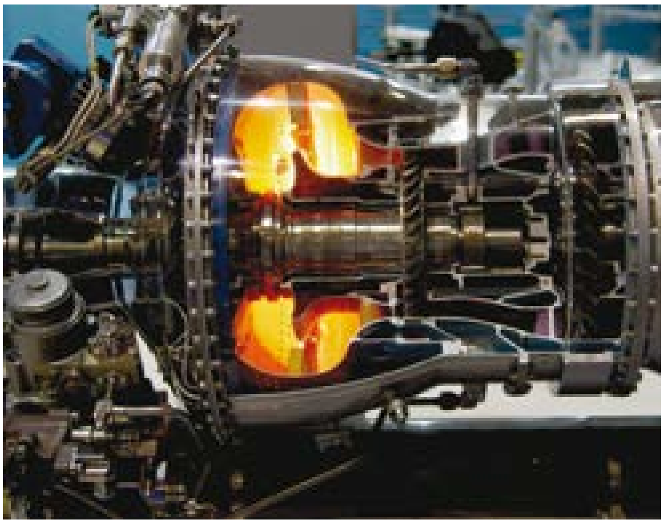


Fig. 2 - Particular of a gas turbine

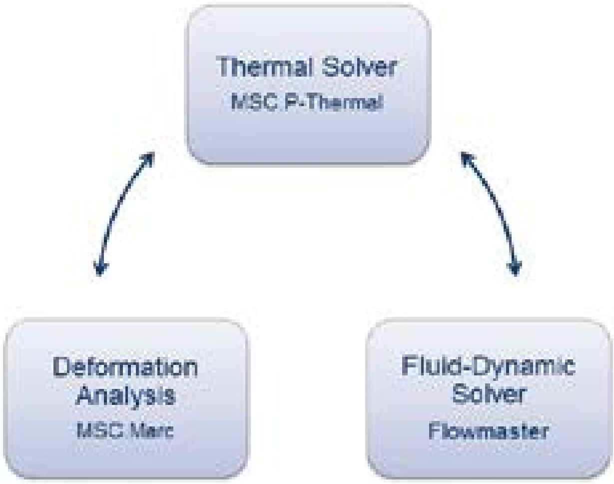


Fig. 1 - Scheme of the integrated multi-physics approach

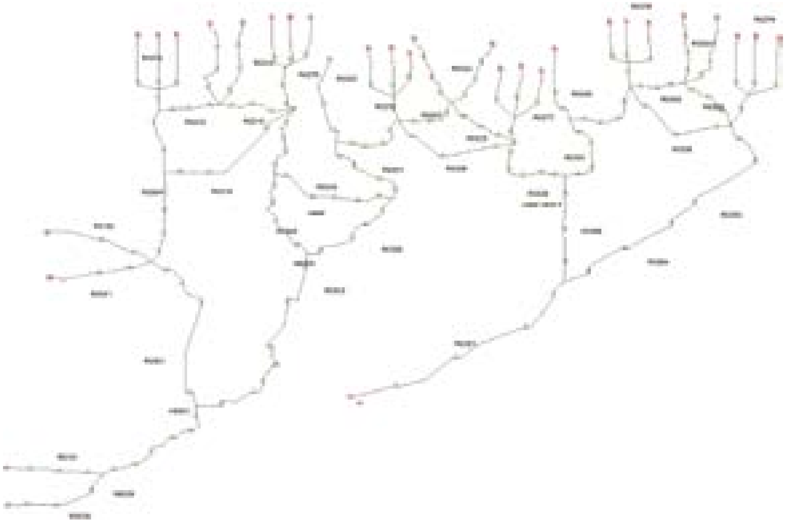


Fig. 3 - Example of a Flowmaster network for the system level fluid-dynamic simulations

The Multi-Physics Approach

The numerical multi-physics approach used to simulate aeronautic engines consists is an automatic procedure developed by AVIO with the help of EnginSoft, as fluid-dynamic solver integrator, capable of managing the operation and data transfer of three different commercial software:

- MSC.P-thermal: thermal solver used for the computation of the temperature distribution in the solid structure;
- MSC.Marc: deformation analysis tool used for the computation of solid structure deformation;
- Flowmaster: system level fluid-dynamic solver used for the computation of the flow field through the gas turbine.

The multi-physics simulation is driven by a specific FORTRAN library implemented into MSC.P-thermal. This library manages the co-simulation by invoking the fluid-dynamic solver (Flowmaster) and the deformation analysis software (MSC.Marc) when required. In particular, the call to Flowmaster is managed by a coupling interface procedure implemented ad hoc in Visual Basic. The coupling interface manages the data transfer between the two codes and manages the fluid-dynamic simulation in all its parts by setting simulation and component data, running the simulation and exporting the results.

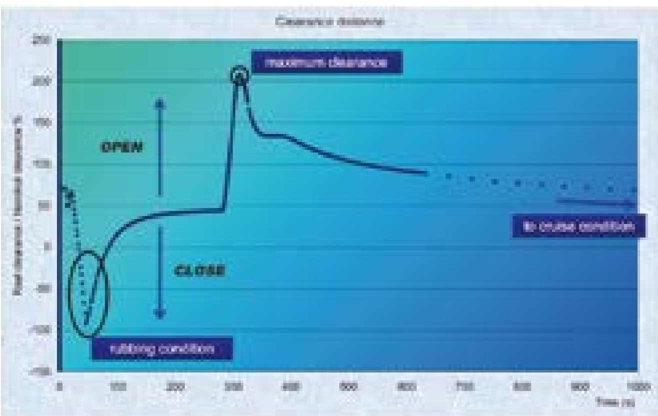


Fig. 4 - Results of an integrated multi-physics simulation of the entire engine mission: clearance (%)

System Level Simulations

To guarantee the required design accuracy, the entire system needs to be modeled, integrating rotor and static systems of the entire turbine. In particular all secondary air systems, cooling circuits, active clearance control devices and the main flow path are to be considered in a system level analysis.

A simulation of the entire engine mission considers idle, take off, cruise, approach and landing phases. A complete simulation lasts about one week and requires about 5000 Flowmaster simulations and about 3000 deformation analyses. All simulations and data transfer are automatically controlled and managed by the thermal solver through the automatic interface procedure as described above.

Conclusions

The implemented numerical multi-physics approach allows to achieve a better understanding of the thermal behavior of the turbine during the entire engine mission early in the design phase. This, in turn, allows to define the optimal geometries, materials and cooling mass flows for active clearance control. In the final analysis, the implemented multi-physics integrated approach, when used early in the design phase, allows to define the optimal clearances capable to achieve high efficiency and, as a consequence, capable to reduce the fuel consumption and to meet the future emission limits.

A. Deponti, EnginSoft,
D. Coutandin, M. Pavan AVIO Group

For more information:
Alberto Deponti, EnginSoft
info@enginsoft.it

Avio is a world leader in the design, development and manufacturing of aerospace propulsion components and systems for both civil and military aircraft. Avio works through the whole life-cycle of the products - from design to maintenance, repair and operations services. Avio is headquartered in Rivalta di Torino, Turin, Italy, and operates across four continents. It employs over 5,200 staff, 4,500 of whom are based in Italy. Avio was founded in 1908 and has played a crucial role in tackling the technological and business challenges of our time. Through continuous investment in R&D, and thanks to its relationships with the top Italian and international universities and research centers, Avio has developed leadership in technology and manufacturing.

BRUCIATORI, FORNI E REATTORI

Maggiore Efficienza e Ridotti Costi di Esercizio con la Simulazione

I processi di combustione sono presenti in diversi settori industriali, dal civile-domestico alla generazione di energia, dall'industria alimentare a quella chimica e petrolchimica, dai trasporti fino a settori fortemente energivori quali l'industria del cemento, del vetro e della ceramica.
Le applicazioni sono molto differenti fra di loro per potenza, per tipologia di combustibile e per sistema di combustione. Nonostante questo la ricerca e lo sviluppo industriale in tutti questi settori hanno obiettivi comuni, sintetizzabili in tre punti:

- riduzione dei consumi di combustibile e delle emissioni di CO_2 tramite incrementi di efficienza dei processi di combustione e di scambio termico;
- riduzione della formazione di inquinanti, soprattutto CO, NO_x e polveri;
- riduzione dei costi tramite l'utilizzo di combustibili alternativi, soprattutto biomassa e combustibili da rifiuti.

Mercato, necessità di efficienza e normative spingono in queste direzioni aziende che producono bruciatori e caldaie di varie dimensioni, reattori, forni e torce.
In questo contesto sempre più aziende affiancano la simulazione termo-fluidodinamica alla sperimentazione. La simulazione viene utilizzata come strumento di "misura virtuale" per entrare in dispositivi in cui le misurazioni sono difficili o impossibili.
La simulazione permette di visualizzare l'andamento dei flussi dei reagenti, il loro mescolamento, l'avanzamento delle reazioni e i fenomeni di scambio termico. La comprensione di questi fenomeni fisici è il presupposto necessario per lo sviluppo del prodotto e il miglioramento del processo.
L'esperienza di EnginSoft in questo ambito copre tutti i settori industriali citati sopra, con applicazioni di varia complessità, che spaziano dallo studio e ottimizzazione dei sistemi di distribuzione aria, alla simulazione dei fenomeni di reazione chimica, di scambio termico e di formazione degli inquinanti. Il software ANSYS CFD permette di simulare la combustione di sostanze gassose, liquide e solide con modelli specifici per le biomasse.
Di seguito alcuni processi e applicazioni industriali che sono stati studiati ed ottimizzati da EnginSoft:

- bruciatori e camere di combustione di varie potenze;
- bruciatori poli-combustibili a gas naturale e olio;
- forni di reforming;
- reattori termici per impianti Claus;
- bruciatori per incenerimento di scrubbed gas;
- reattori per abbattimento CO;
- forni da cementeria;
- forni per cottura di prodotti ceramici quali piastrelle, prodotti estrusi e stoviglieria;
- forni per trattamento termico;
- sistemi per recupero di calore e generazione di energia da biomassa;
- sostituzione di combustibili convenzionali con combustibili da rifiuti.

Un primo esempio è lo studio condotto da EnginSoft insieme a Siirtec Nigi su un reattore chimico per l'ossidazione parziale di NH_3 ed H_2S.
Siirtec Nigi è un'azienda leader nell'industria oil&gas, di raffinazione e petrolchimica. Know-how ed esperienza decennale, con più di 500 impianti in 40 paesi nel mondo, fanno di Siirtec Nigi un riferimento per i processi di trattamento dei gas acidi, recupero e degassing di zolfo, incenerimento e tail gas clean-up.
Il sistema studiato ed ottimizzato è un reattore già operativo e di cui si vuole verificare il comportamento per uno scenario di esercizio con portate differenti rispetto a quelle di progetto. In particolare si vuole visualizzare l'andamento dei flussi in uscita dal bruciatore, calcolare il grado di mescolamento dei reagenti, il completamento delle reazioni e il conseguente campo di temperatura nella camera di combustione. Obiettivo dello studio è garantire il massimo grado di reazione di NH_3 e H_2S.

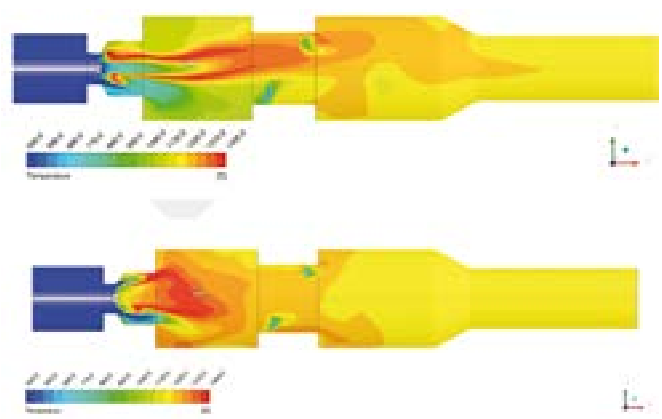


Fig. 1 - campo di temperatura nel reattore originario (sopra) e in quello modificato (sotto)

L'analisi della configurazione originaria del sistema ha messo in evidenza come una variazione delle portate rispetto a quelle di progetto provoca una lunghezza eccessiva della zona di combustione.
La visualizzazione dei campi di velocità e delle varie specie chimiche ha fornito indicazioni sulle modifiche da apportare alla testa del bruciatore. Le modifiche apportate hanno permesso di migliorare il mescolamento delle specie e di ottenere una migliore combustione nella parte anteriore del reattore. Il beneficio finale è stato un maggiore completamento delle reazioni e l'abbattimento di NH_3 e H_2S in uscita dal reattore.
Figura 1 mostra l'effetto delle modifiche del bruciatore sul campo di temperatura e sulla lunghezza della zona di combustione.

Un secondo esempio è lo studio condotto da EnginSoft insieme a SITI B&T Group su un forno per la cottura di prodotti ceramici.
SITI-B&T Group rappresenta un marchio che da cinquant'anni contraddistingue e rappresenta una gamma di prodotti e servizi, nel settore delle macchine per la produzione di piastrelle in ceramica, dei sanitari e della robotica avanzata, progettati e realizzati per soddisfare le esigenze di ogni tipologia di cliente.

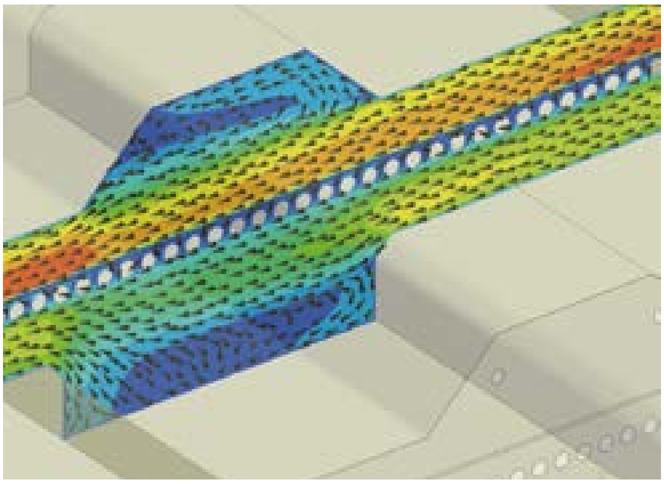


Fig. 2 - campo di velocità nella nuova configurazione del forno per cottura di piastrelle

"Il valore della ricerca – afferma l'ing. Casoni di SITI - garantisce l'alto livello di qualità dei propri prodotti. Siamo convinti che investendo in ricerca e sviluppo, siamo in grado di offrire nuove tecnologie che di fatto rappresentano il nostro motore di crescita".
Un forno per la cottura di piastrelle è un sistema di notevoli dimensioni e complessità, con alcune decine di metri di lunghezza e oltre 70 bruciatori a gas naturale. Le piastrelle entrano da un lato del forno, scorrono su rulli e vengono esposte al calore prodotto dai bruciatori posti sopra e sotto i rulli. La potenza utilizzata è di circa 2 MW. Per una corretta cottura delle piastrelle e quindi per ottenere una buona qualità del prodotto finito è necessario che la piastrella segua una ben definita storia di temperatura tra ingresso e uscita forno. La variabilità di impasto della piastrella richiede una forte flessibilità della macchina termica. Lo scopo dello studio condotto con SITI B&T Group è comprendere se una modifica della volta del forno permette di mantenere la stessa qualità del prodotto finito riducendo la spesa energetica del 25%. In particolare si vuole verificare se una volta a sezione variabile permette di ottenere un migliore scambio di calore tra gas e piastrelle riducendo quindi il fabbisogno di combustibile.
Il modello fluidodinamico riproduce due moduli di pre-riscaldo e sedici moduli di cottura, per una lunghezza complessiva di circa 40 m. Lo studio comparativo tra il forno a volta costante e quello a volta variabile ha messo in evidenza che con la volta variabile si ottiene un notevole miglioramento dello scambio termico. La piastrella segue la stessa storia di temperatura pur riducendo il fabbisogno energetico del 25%. Non solo, il campo di velocità dei gas e la temperatura, cioè la cottura, delle piastrelle sono più uniformi. Le simulazioni CFD hanno anche permesso di calcolare le dispersioni termiche dalle pareti del forno, fornendo così indicazioni per un ulteriore miglioramento energetico e abbattimento dei costi di esercizio dell'impianto.

"Grazie alla simulazione abbiamo verificato correttezza ed attendibilità delle intuizioni progettuali del forno. Le informazioni ottenute, sui fenomeni studiati e il comportamento dello scambio termico forno/piastrella, si sino rivelate fondamentali e utili per questo ma anche per altri progetti di ricerca e sviluppo. L'incremento di prestazioni e abbattimento della spesa energetica sono tra i principali vantaggi conseguiti attraverso l'attività CAE" - Ing. Casoni di SITI B&T

Per maggiori informazioni:
Massimo Galbiati - EnginSoft
info@enginsoft.it

Green light to high speed trains

Bombardier achieves the best design to minimize energy consumption with modeFRONTIER

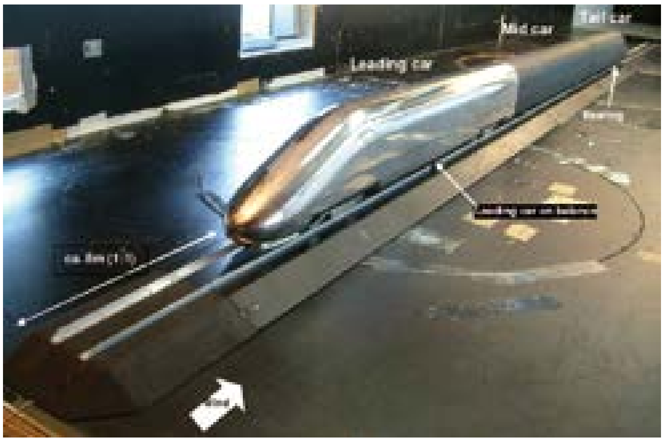


Fig. 1 - General view of drag measurement

“Rail transportation is a concrete eco-friendly solution for sustainable mobility. Therefore, if we decrease the aerodynamic resistance of our trains, we can increase energy efficiency and further reduce CO2 emissions”, says Alexander Orellano, Head of Aerodynamics at Bombardier, world leader company in aerospace and rail transportation. Reducing energy consumption implies optimizing the aerodynamic shape of a vehicle, thus inevitably facing two main opposing factors: the best models for drag do not have a good crosswind stability, and vice-versa. In addition, high passenger capacity conflicts with optimal aerodynamic shape and elegance and functionality not always go hand in hand. These are some of the reasons that made Bombardier choose modeFRONTIER, the multi-objective, multidisciplinary optimization platform, for their award-winning ZEFIRO 380 train design. “The application of the multi-objective optimization method to Bombardier high-speed trains leads to a highly competitive product, entailing both to energy efficiency and cost reduction, due to a lower traction power dimensioning”, declared Mr. Orellano. The goal was to find a Pareto-optimal, or trade-off, design which would simultaneously give low drag and good cross-wind stability characteristics. The solution was obtained using modeFRONTIER, not only to integrate the various CAE tools in use at Bombardier, but also to drive the geometry modification and simulation process providing the necessary graphical tools for the statistical interpretation of results. modeFRONTIER, uses genetic algorithms to determine Pareto optimal solutions, combining 3D models and simulations of aerodynamic drag and cross-wind stability. Bombardier experts considered as many as sixty different design parameters in the modeling phase, taking into account the train’s outer shell, cab, crash structure and ergonomic constraints.

The company was therefore able to reduce the aerodynamic resistance by 20%, obtaining a decrease of energy consumption of about 10%. By using modeFRONTIER, Bombardier engineers were able to choose from a selection of designs in order to suit particular styling preferences, but secure in the knowledge that each complies with the principles of optimized energy performance and maximum stability and safety.

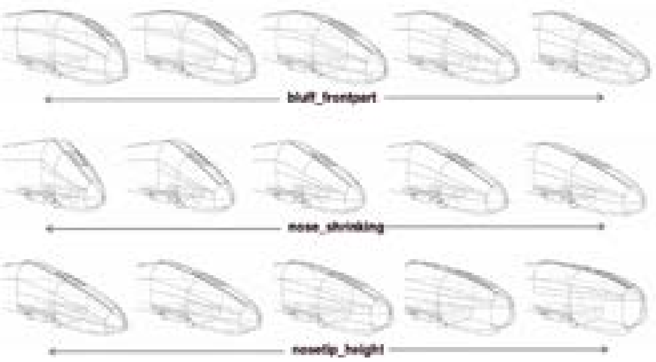


Fig. 2 - Example of model variability (3 parameters out of 60)

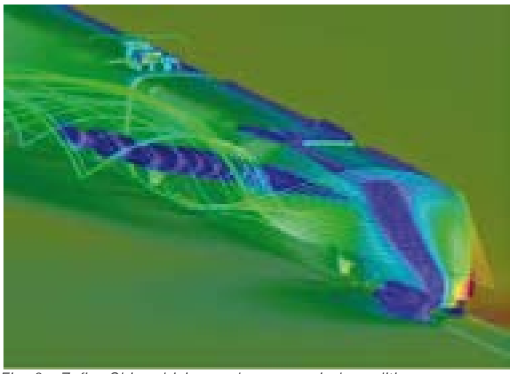


Fig. 3 - Zefiro China driving under cross wind conditions.

Bombardier is a global transportation company, present in more than sixty countries on five continents, which designs, manufactures, sells and supports a wide range of products in the aerospace and rail-transportation sectors. Bombardier is headquartered in Montréal, Canada, and operates in large part of the world. The company is setting a new benchmark in Very High Speed rail travel with the “Bombardier Zefiro” train, the world’s most economical and eco-friendly VHS train, which reaches an operating speed of up to 380 km/h. For more information visit: www.bombardier.com

BOMBARDIER

Modeling application to optimize a two stroke engine at 3000rpm

In this article an industrial application of numerical modeling is presented. Abamotor Energía s.l. is leading the project to improve its two stroke petrol engine. Nowadays this engine works well above 4000rpm, and they want to improve the performance and to reduce the vibrations and noise at low speed, particularly at 3000rpm for some applications. To reach this objective, simulation software of wave dynamics developed by CMT will be used.

Introduction

In this article an industrial application of numerical modeling is presented. The performance of a two stroke petrol engine (Minsel M165 model) needs to be improved at low speed, since lower speed means less vibrations and noise for the user of portable machines (mowers, rail grinding machines or agricultural machinery), where it is typically applied because of the high power-to-weight ratio of this type of engines. This particular engine is originally works above 4000rpm and gas emissions are typically higher at low speed than at high speed, so with this improvement it will be easier to accomplish future regulations.

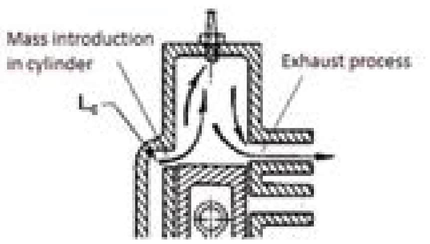


Fig. 1 - Piston controlled inlet port, the simplest of the designs and the most common in small two stroke engines

It has been developed a numerical model of the M165 engine in order to optimize their performance at low speed (3000rpm), changing the charge renewal system, reducing the short circuit of the mixture and improving the filling of cylinder, where the geometry of the exhaust and transfer port, and the exhaust pipe have great influence. This engine charge renewal is piston-controlled inlet port (Fig.1) and has Schnürl loop scavenging (Fig 2).

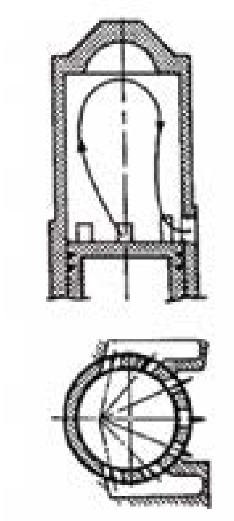


Fig. 2 - Schnürl loop scavenging prevents the fuel/air mixture from traveling directly out the exhaust port, but also creates a swirling turbulence which improves combustion efficiency, power and economy

This behavior is difficult to predict analytically, and the design of these engines is widely dependent of experimental trials. This dependence is reduced introducing numerical modeling, since it predicts the influence of different variables in the process and the engine global behavior after some changes fast and economically.

The methodology used to develop the project was:

1. Characterization of the engine.
2. Preliminary 1D model of the complete geometry of the engine.
3. Experimental tests to compare the model and actual working conditions.
4. Model validation.
5. Redesign components for engine optimization.

1D model: Finite Amplitude Wave Action Model

The software developed by CMT OpenWAM© is capable to model two stroke petrol engines and manage thermo-fluid dynamic processes in the circulating flow across the engine, including heat transmission, flow momentum variation or flow friction against the pipes. These equations are dependent of time variable and spatial dimensions, but it's considered that the variation is only dependent of one spatial variable.

Consequently one-dimensional compressible flow equations are applied in pipes, crankcase or cylinder for each partition:

1. Continuity equation
2. Equation of angular momentum conservation
3. First law of thermodynamics

$$\frac{\partial \rho}{\partial t} + \rho\frac{\partial u}{\partial x} + u\frac{\partial \rho}{\partial x} + \frac{\rho u}{F}\frac{dF}{dX} = 0$$

ρ = Density, u = Speed, p = Pressure, F = Force, x = Position, t = Time

$$\frac{\partial}{\partial x}(pF) + p\frac{dF}{dx} - \rho K = \frac{\partial}{\partial t}(\rho F u) + \frac{\partial}{\partial x}(\rho F u^2)$$

Where: $K = \frac{f \rho u^2}{2}\pi D$

$$q\rho F = \frac{\partial}{\partial t}\left[(\rho F)\left(C_v T + \frac{u^2}{2}\right)\right] + \frac{\partial}{\partial x}(\rho u F)$$

T = Temperature, q = Heat source, C = Specific heat

The approach of these equations in each partition of the ducts is a system of partial differential equations that are solved by the finite difference method, developed by Lax-Wendroff.

To complete the model, the necessary inputs have to be inserted to the software:

1. Execution data: nº of cycles, partition type, resolution method…
2. Ambient conditions: pressure, temperature, humidity.
3. Operating conditions: engine speed, fuel consumption, fuel heat capacity (HLV).
4. Geometrical parameters: diameters, lengths, angles and thicknesses of different elements.
5. Characteristics of each element: roughness, number of partition.
6. Heat transmission data: initial temperature, specific heat.
7. Discharge coefficients (Cd): ratio between real and isentropic flux mass. In duct connections for example between a pipe and a tank, the real flux mass will be different from an isentropic one, and it is usual to represent this behavior with this coefficient. Values are available in literature and are a useful experience, but also, it is possible to measure them experimentally.
8. Combustion law: to define the burned mass fraction (FQL) Wiebe law (4) is used. The necessary parameters are the degree of completeness (C), the combustion angle extension ($\Delta\alpha c$), the combustion start point ($\alpha 0$) and a shape parameter (m). Experimental data are necessary to achieve a good adjustment of parameters as further explained.

$$FQL(\alpha) = 1 - exp\left[-\alpha\left(\frac{\alpha - \alpha_0}{\Delta\alpha_c}\right)^m\right]$$

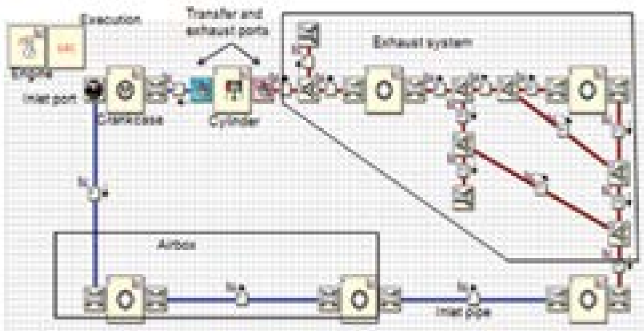


Fig. 3 - Engine model view with elements: Cylinder, crankcase, exhaust system, inlet, transfer and exhaust port, and extra pipes and connections.

After launching the simulation, the outputs of the model are for each element and each partition:

1. Average results: Temperature, pressure, flow, power.
2. Temperature, pressure, mass flow, volume, mass and % burned in cylinder.

Fig. 4 - Pressure sensor layout, hot wire anemometer and consumption measurement

Figure 3 presents the interface of the engine model, where are shown every basic element of the engine and include the described necessary input data for each one.

The most complex is the exhaust system, what is a combination of tanks and holey pipes for diminution of sound but has also importance in performance.

Experimental test for validation

Despite numerical modeling minimizes dependence on experimental, it is necessary to validate the model in order to control the influence of used simplifications.

The measured variables have the objective to check heat transfer, combustion or the pressure waves inside the engine are:

- Temperature: at exhaust pipe, with thermocouple type K.
- Pressure: at cylinder, exhaust pipe (piezoelectric) and crankcase (Piezoresistive).

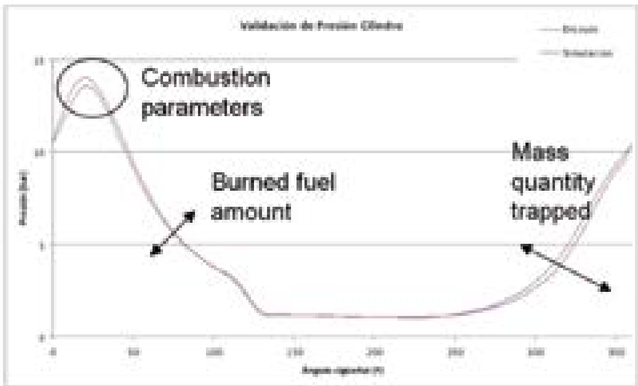


Fig. 5 - Simulation-experimental comparison for combustion chamber pressure at 3000rpm

- Air flow: at the inlet port with a "hot wire anemometer".
- Consumption: measuring the time to consume a quantity of fuel.

Model validation

In this context, validation means adjustment of inputs (discharge coefficients or combustion parameters) to have the same results between experimental trials and the model. This is made for different speeds to be as general as possible but always with the accelerator full opened. After this adjustment, it is reasonable to think that with little changes of the design.

In the article, it is shown the comparison only for the case of 3000rpm in order to represent the grade of proximity that was achieved. Firstly, it is reported the comparison between the output "instant cylinder pressure" during the cycle when already steady state has taken place

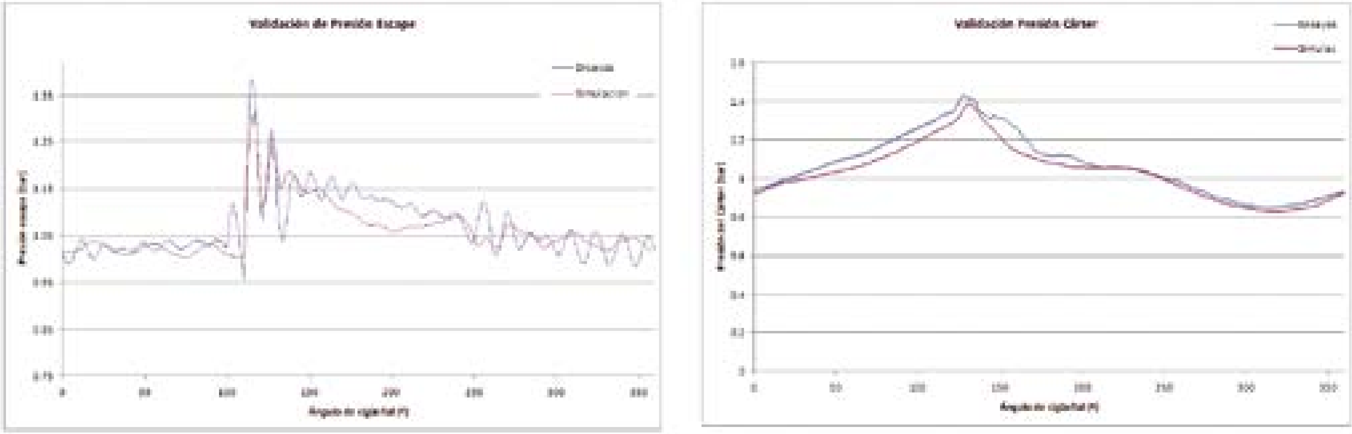


Fig. 6 - Simulation-experimental comparison for exhaust port and crankcase pressure

and the piezoelectric transducer measurement. In the first approach, it is really different, but adjusting Wiebe laws parameter and fuel consumption and the air trap in cylinder (Fig. 5) we arrive to have the real behavior inside the cylinder.

The scavenging in a two stroke engine is provoked by the vacuums and pressures in the different parts of the engine, then not only is the cylinder pressure important. In figures 6 and 7 are compared for exhaust and crankcase pressures. To validate this part, the critical parameters are the discharge coefficients, and without specific experimental test for measuring them, it is necessary an iterative process being very difficult to arrive to an exact adjustment.

After these results the air mass flow of simulation is 4,832g/s really close to the 4,835g/s experimentally measured.

Components redesign to optimize the engine:

Cylinder and exhaust pipe geometries

Once the model is reliable, it is useful to see how some variables fluctuate during the cycle in order to get some conclusions about the engine performance. The main goal of the project was to improve the performance, but regarding the variables to maximize or minimize which are the best to optimize?

The decision taken was to maximize the mass quantity trapped in the combustion chamber when it is ignited, because engine power is directly dependent on it. It is necessary to check that it doesn't increase residuals in the previous cycle in chamber, don't have higher consumption and don't have higher mass short-circuited.

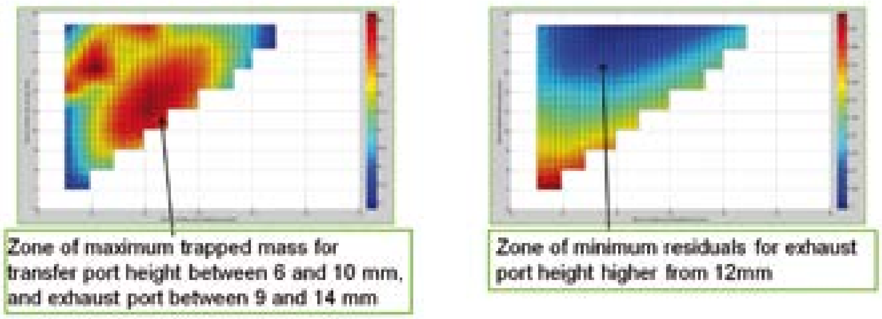


Fig. 7 - Result plots to evaluate the optimum geometry

About the variables for redesign, it was decided to use the geometry of transfer and exhaust port (height and width) and exhaust pipe geometry (lengths, volumes) because both have a great influence regarding the charge renewal.

There is presented an example in figure 7 to explain how works the approach used to optimize. There are changed the heights of exhaust (y), the transfer port (x) and we could see what zones maximize the air trapped. After checking how residuals vary in these zones, the consumption and short-circuit.

This approach is correct but requires very hard handmade work, and the cases capable to analyze are limited. In future work it may be implemented a mathematical programming applied to optimization, since the problem have an analytical function to maximize depending of the variables and have also some restrictions to fulfill.

Conclusions

A reliable model of actual engine validated with experimental tests is obtained reducing the need of experimental test for this and future designs. It's verified that changing elements what have influence in charge renewal could improve the trapped mass till 15% without aggravate emissions, residuals or consumption.

Acknowledgements

The authors would like to acknowledge the Spanish Ministry of Economy and Competitiveness for financing the project TRACE for the transference of research results from CMT to Abamotor Energía s.l.

B. Zabala, A. Igartua M. Conte,
IK4-Tekniker, Eibar, Spain

R. Zabala,
Abamotor Energía, Abadiano, Spain

H. Climent
CMT-Motores Térmicos, Universitat Politècnica de València, Spain

Dynamic analysis of a Variable Displacement Oil Pump shaft under crankshaft torsional vibrations and internal pressure evaluated via CFD

This paper deals with an activity carried out at the Calculation & Simulation Department in Pierburg Pump Technology Italy SpA. The subject of analysis is a vane oil pump for truck applications. This product represents new generation oil pump in automotive industry thanks to the possibility to reduce the displacement at high engine speed for fuel save consumption. Aim of this study is the calculation of dynamic torque applied to pump shaft in order to have a better understanding about the causes of unfitting failures between rotor and pump shaft revealed in aggressive durability tests.

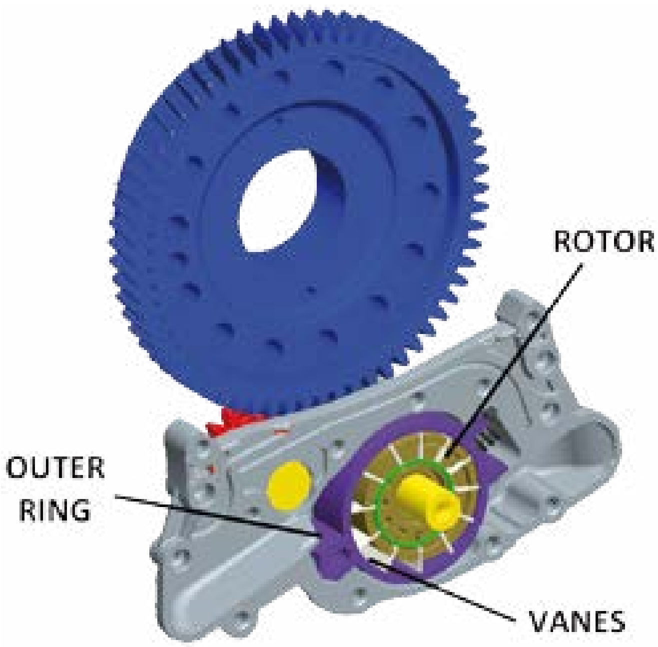


Fig. 1 - Assembly (rear view)

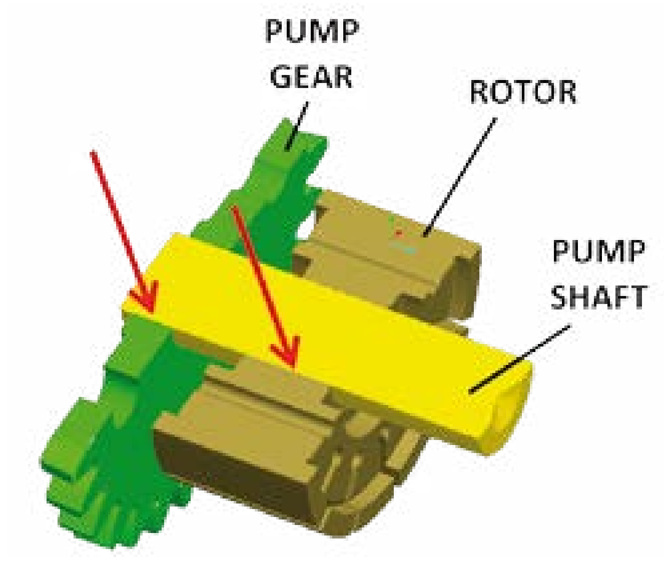


Fig. 2 - Pressfitted areas

Product presentation

Variable displacement oil pumps represent a significant contribution for fuel saving capability in automotive industry. In comparison to conventional gerotor pumps, they have the possibility to optimize the oil flow according to engine demand, with a significant reduction of power absorption. The oil flow rate of a vane pump, such as that of a general volumetric pump, depends on its actual displacement, i.e. the difference between the maximum and minimum trapped volumes. This difference is a function of the pump eccentricity that is defined as the distance between the rotor axis (which is fixed) and the control ring axis. That's why, in order to obtain the displacement variation in this kind of pumps, the control ring is made to slide in housing.

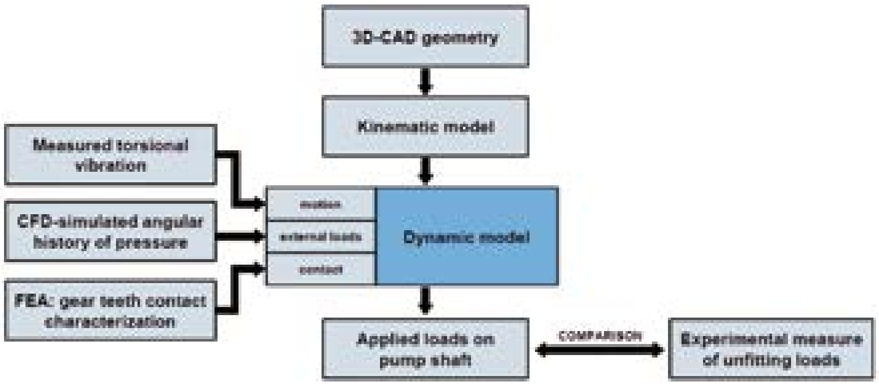


Fig. 3 - Workflow

The variable displacement oil pump considered in this study is driven by a train of three gears (Figure 1): engine gear (in blue), idle gear (in red) and pump gear (in green). Both pump gear and rotor are fixed to pump shaft by two different press fitting areas (Figure 2).

Procedure

In order to reach the target of calculation (dynamic torque on pump shaft) a multibody model was created taking care about driving action by crankshaft and braking actions due to pressure distribution inside the pump itself. The entire geometry has been imported by existing 3D-CAD model. Crankshaft motion has been imposed including its torsional vibrations, which is a fast dynamic function. This required a dedicated experimental activity on instrumented engine. Forces elements have been applied on each vane as a function of angular position. Angular history of hydraulic pressure has been calculated by means of CFD-simulation in working conditions where uncoupling between rotor and shaft has been observed during durability tests.

Setting of contact parameters is critical in this kind of analysis because the response of system is highly influenced by this. In this work contact parameters have been set up basing on a dedicated static structural FEM simulation where stiffness of gear contact has been evaluated. Scheme in Figure 3 shows the workflow of the entire activity. Aim of this study is the calculation of applied dynamic loads on pump shaft under crankshaft torsional vibration and internal hydraulic pressure evaluated via CFD. Resulting maximum instantaneous loads will be compared to ones obtained on experimental tests (see the specific paragraph) in order to understand if the dynamic of the system can explain by itself the reason of uncoupling failure between rotor and shaft.

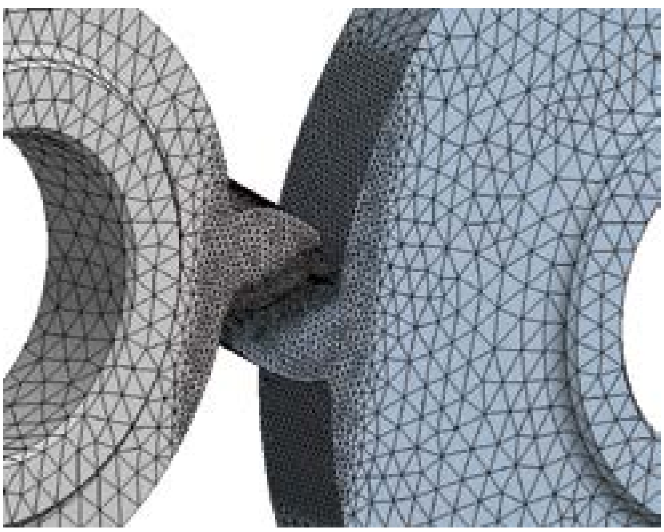


Fig. 4 - Single-tooth contact in idle and pump gears

Data acquisition

The implementation of crankshaft torsional vibrations into dynamic multibody model required dedicated experimental activity on engine that has been conducted by the customer. Full map of acquisition data has been collected on engine instrumented with employed sensors. Real rotation of crankshaft has been measured with respect to the ideal rotation. Ideal rotation is the imaginary position that crankshaft should have moving at constant speed.

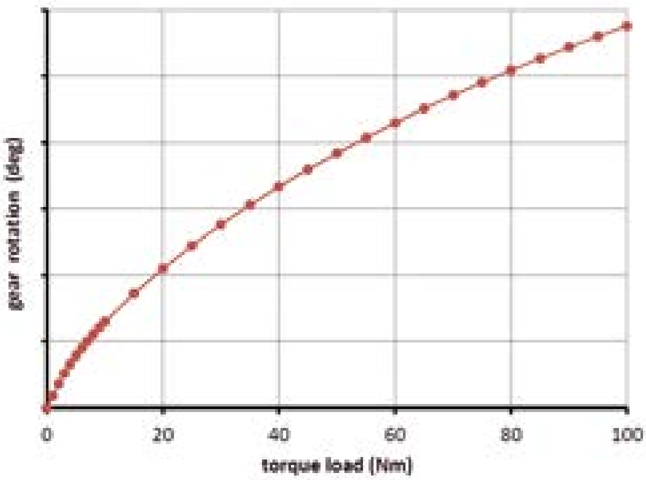


Fig. 5 - Resulting contact stiffness

FEM model

FE analysis has been conducted in order to numerically evaluate the stiffness properties on contact tooth-tooth. Target is to obtain the existing law between contact force and gear axis rotation, as characterization of contact stiffness. Special features have been built in order to obtain idle gear and pump gear with one single tooth (Figure 4). This choice has been done in order to guarantee a unique contact between a couple of teeth. In this model pump gear axis has been fixed-constrained and idle gear axis has been left free to rotate around its axis. A ramp of crescent torque up to 100 Nm has been applied to idle gear. The relative axis rotation has been post processed.

The resulting contact law is shown in Figure 5. Stiffness behavior calculated by FE analysis has been reproduced in multibody environment. Obviously, stiffness behavior in FE analysis uses a different approach with respect to stiffness behavior in multibody

Fig. 6 - All oil volumes needed for CFD

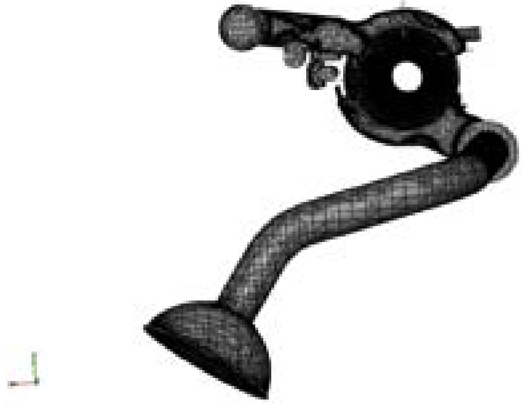


Fig. 7 - Mesh of the pump oil volumes

software that on the contrary uses rigid bodies. Despite of this, it is important that the law between contact force and axis rotation has been maintained.

CFD model

The CFD simulation has been completed in Simerics Pumplinx environment. To setup the model, four distinct oil volumes have been extracted with a Boolean operation of subtraction: inlet volume, outlet volume, suction pipe volume and rotor chambers volume. Result of the operation is presented in Figure 6.
Importing as STL files these volumes into Pumplinx software, the final CFD mesh has been created for the analysis (Figure 7). A realistic axial clearance has been simulated in the final mesh in order to suit the simulation closer to reality. CFD analysis has been setup in the requested engine operating condition at high temperature representing the performed durability tests during which the uncoupling between shaft and rotor has been observed.

To suit the simulation closer to reality for the durability test on engine, a defined aeration level of oil has been simulated. In fact because of the presence of air and/or vapor bubbles in oil, bubbles can implode due to the increase of local pressure when a rotor chamber is connecting to delivery side. CFD simulation has been set up as a transient analysis with moving mesh. Hexahedral elements have been used because they are most efficient elements for long thin geometries. In addition, it is the most efficient flexible mesh in case of needed mesh deformation to allow a better squeezing of the elements due to the rotation of the rotor. A monitor point has been placed in one rotor chamber and, because rotor is rotating, the point is rotating with it. After few initial iterations to reach convergence, variation of pressure and gas volume in the pocket with the rotation angle can be post-processed. This CFD derived pressure load has been used as input data into the following multibody model of the pump.

Multibody model

A multibody model of the pump has been implemented (Figure 8). Inertia properties for the bodies have been obtained on the basis of 3D CAD drawings and technical data.

By geometrical point of view, the case of medium backlash has been modeled. Each gear has been connected to chassis by revolute joints. Rotational degree of freedom only is allowed for each gear. The kinematic of multibody model has been built focusing on the reproduction of torsional behavior of the real mechanism. Gear transmission has been guaranteed by solid contact elements; they consist on an equivalent spring-damper element which law is based on overlap volume due to rigid teeth penetration. Preliminary multibody model has been built in order to set up contact parameters. In this model special features of single-tooth gears (Figure 4) have been imported as in the previous FE analysis. A loop of iterative simulations has been run until the

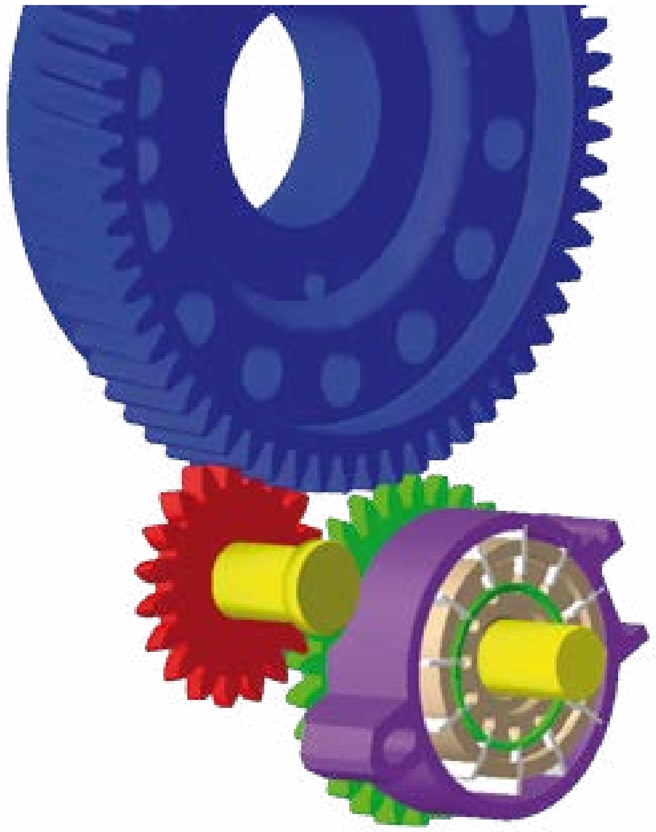
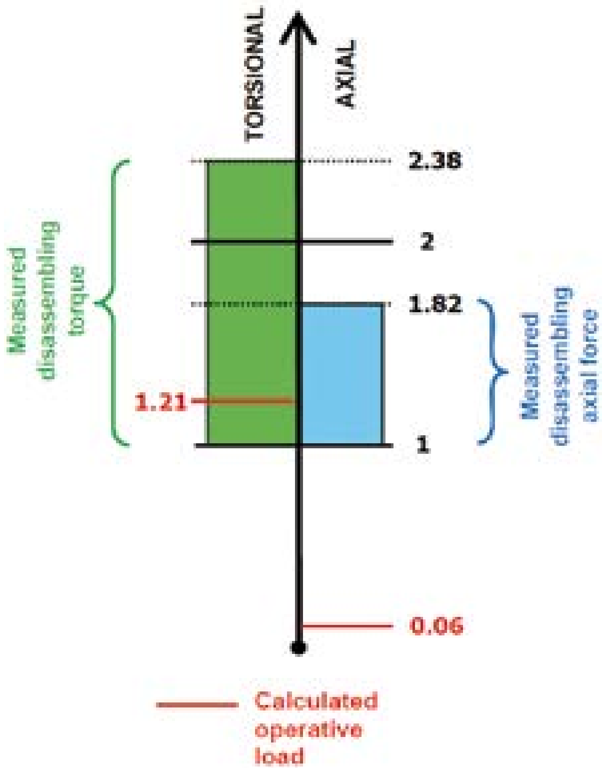


Fig. 8 - Multi-body model

Fig. 9 - rotor decoupling

same stiffness law in Figure 5 has been matched. Once the set up phase is terminated, the contact settings have been frozen and maintained in subsequent simulations where gears realize multiple contact. Hydraulic pressure effect has been modeled by force elements applied on each vane. Pressure level depends on rotor angular position and, as previously explained, has been calculated via CFD-simulation. In this analysis additional torsional loads have been modeled (representing frictional and viscous effects) in order to match experimental adsorption data.



Experimental tests

Experimental activity has been conducted in PPT testing department. Special fixture has been built in order to measure the disassembling torque and disassembling force between rotor and shaft under hydraulic press load. Obviously the unfitting loads are dependent by the real interference level on the coupling rotor-shaft. Extreme loads of the range refer to the condition of max and min interference. In the following table the minimum level is normalized to one:

Measured disassembling torque
1 ÷ 2.38 (normalized values)
Measured disassembling axial force
1 ÷ 1.82 (normalized values)

Results

Several events of rotor unfitting occurred during severe endurance tests on engine as shown in Figure 9. In this work it has been built a multidisciplinary model in order to evaluate if considered external actions themselves can explain the causes of rotor disassembling. Forcing actions on dynamic system are mainly represented by crankshaft torsional vibration and hydraulic pressure. By the frequency content analysis of dynamic response it has been possible to assert that in this specific case the dynamic imposed by hydraulic internal pressure is dominant with respect to crankshaft vibrations.

At least, simulation result of max instantaneous torque and axial force between rotor and shaft has been post processed. In the following table the minimum level is normalized to one:

Calculated torque 1.21 (normalized value)
Calculated axial force 0.06 (normalized value)

Calculated loads have been compared to experimental data both in torsional direction (range 1÷2.38) and axial direction (range 1÷1.82). Calculated max instantaneous torque is higher than the minimum measured value (1.21 > 1). So it is probable that considered external actions (torsional vibrations and hydraulic pressure profile) are sufficient to explain the disassembling of rotor from its axis. Considering axial direction, calculated force is not critical because it is much lower than measured one (0.06 << 1). So in axial direction modeled loads are not sufficient to explain rotor unfitting.

Conclusions

In the application presented in this paper and for considered working conditions, the contribution of hydraulic internal pressure on the dynamic of system is dominant with respect to the contribution due to the presence of crankshaft torsional vibrations. Main result of the whole multidisciplinary activity is that calculated operating load of rotor-shaft coupling is internal to the range of measured disassembling load, so it is possible that rotor disassembling occurs due to specific operating conditions.

Torsional loads are critical, axial loads are not critical in this application. Since in real parts also axial movement of rotor were found, by results this evidence can be explained as a secondary effect that occurs after torsional disassembling.

Andrea Barbetti, Matteo Gasperini, Raffaele Squarcini
Pierburg Pump Technology Italy,
Calculation & Simulation department

For more information:
Valentina Peselli - EnginSoft
info@enginsoft.it

AIR INTAKE MANIFOLD
Design and Optimization

Magneti Marelli Powertrain is a Magneti Marelli business dedicated to the production of engines and transmission components for cars, motorbikes and light vehicles. Magneti Marelli designs and produces engine control systems for gasoline, diesel and multi-fuel engines, including electronic Control Units, Pumps and Injectors for PFI and GDI engines, Air Intake Manifolds, Throttle Bodies, Automated Manual Transmissions and Dual Clutch Transmissions, as well as Systems and Components for Hybrid and Electric Drives.

The design process for an air intake manifold has been tackled throughout three phases. The first stage represents and starts from the customer request. The second is to identify the design objectives that will drive the 1D virtual tests of the engine. In the end, by taking advantages from the design experiences of the first two steps, we could implement a novel, faster, and a more efficient method to meet the customer's requirements.

Product Engineering Engine Design	S.o.R.	Edition 1	
	Document Name:	Revision 01	Date 30/01/2012
Component / System Name FUEL RAIL	Engine:	Comp. Code xxxx	Application xxxx

4. Engine general data (A engine: 1.6 NA - B engine: 1.8 NA)

Displacement:	A: 1.598 [cc] B: 1.747 [cc]	Max Power @ engine speed:	A: 117 [HP] @ 5500 [rpm] B: 130 [HP] @ 5250 [rpm]
N. of cylinders:	4	Max Torque @ engine speed:	A: 168 [Nm] @ 4000 [rpm] B: 188 [Nm] @ 4500 [rpm]

Fig. 1 - Typical specification request from engine manufacturer

Just like the project we summarize in this article, a typical challenge that we get from our customers (in this case an engine manufacturer) is that a new air intake manifold has to satisfy specific requirements. As shown in Figure 1, the requests includes the development of a 1.8 liter naturally aspirated engine featuring the power of 130 HP at 5250 rpm, with a Torque of 188 Nm at 4500 rpm.

To satisfy these design specifications, a virtual prototyping method has been applied. GT-Power (from Gamma Technologies), as one of the tools for the virtual engine and powertrain platform, is the analysis software used in the project for the 1D simulation of the engine and vehicle. GT-Power is used in the early stages

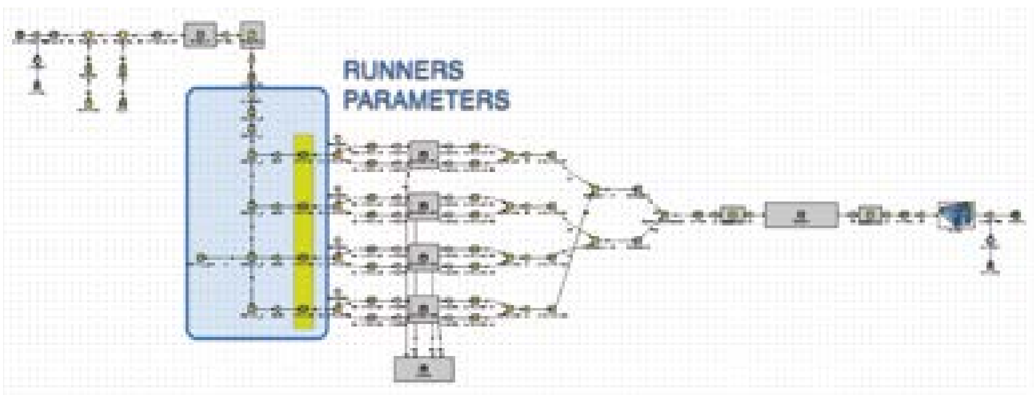


Fig. 2 - GT-Power Engine 1D schematic representation. The blue box shows the geometric parameters of the intake runners

Fig. 3 - Three-dimensional model of intake runners (LEFT) and one-dimensional GT-Power model of the same runners (RIGHT)

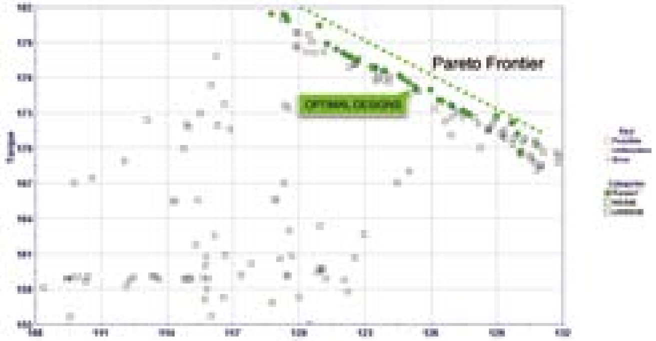


Fig. 5 - Targets scatter chart with brake Power (x-axis) and brake Torque (y-axis)

of the engine development. However, an analysis software is not a sufficient solution for achieving the objectives included in the specifications. It does provide spot analyses though which is the key design task for the engineer in the search of the optimum. For this purpose, also modeFRONTIER has been adopted as multi-objective optimization software tool.
The GT-Power engine model is a numerical artifact that virtualizes the real behavior of the actual engine. More specifically, the 1D

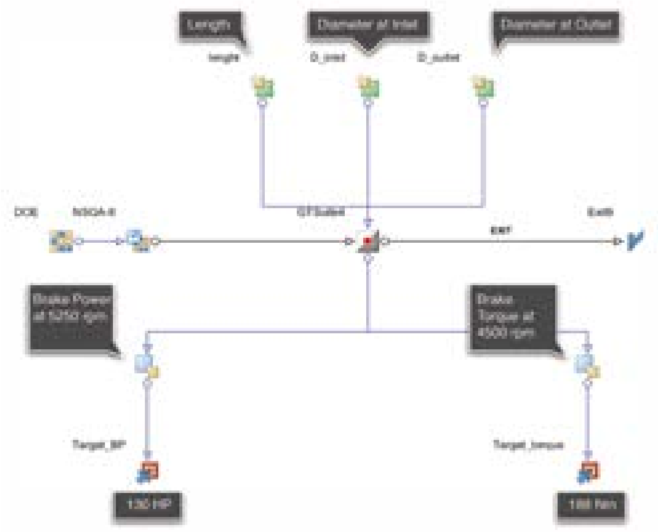


Fig. 4 - modeFRONTIER workflow

model simplifies the representation of such a model by reducing the real engine characteristics to lumped parameters (see Figure 2). In this article, only the geometrical entities are used as free parameters for designing the manifold. In fact, the runners dimensioning is the main goal of the optimization process.

Therefore, the actual 3D manifold geometry is "reduced" to a schematic representation (1D) as shown in Figure 3, where the parameters that define the design space are:

- Length of the runner
- Diameter of the runner's inlet
- Diameter of the runner's outlet

These three parameters are the only input variables to be set in the modeFRONTIER workflow. The GT-Power node is the only analysis

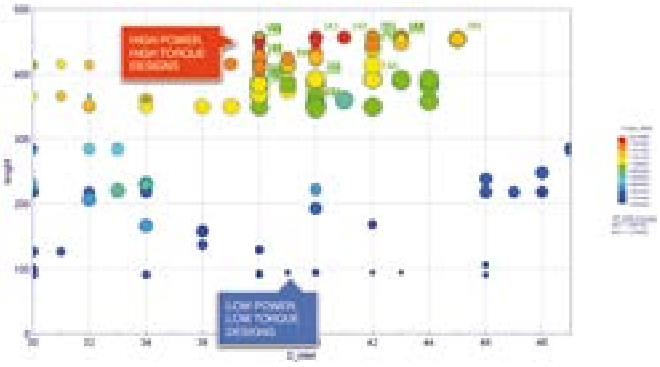


Fig. 6 - 4 dimension scatter chart with runner's diameter at Inlet (x-axis), runner's length (y-axis), brake Torque as color, and brake Power bubble diameter. Green labeled are the Pareto designs

to be run. Brake power at 5250 rpm and Brake torque at 4500 rpm are set as output values that must meet the design goals defined by the customer, such as obtaining a power of 130 HP and a torque of 188 Nm. As also reflected in the modeFRONTIER diagram in Figure 4, Power target and Torque target are the objectives of the optimization. In order to achieve both targets, a multi-objective optimization algorithm is chosen, the NSGA-II.

The results of the first optimization loop are plotted in the scatter chart in Figure 5. The chart presents the brake power on the x-axis and the brake torque on the y-axis. The square points are the simulated manifold configurations. The green circles are the individuals belonging to the Pareto frontier (the optimal designs).

It is important to understand if the optimization has achieved the design goals in order to possibly improve existing solutions. The plot in Figure 5 tells us how the optimizers strive for improving both outputs. It is equally important to see how the input parameters behaved during the optimization process. As illustrated in Figure 6, the bubble chart displays the runner's diameter at inlet and the runner's length. It also reflects the designs where the color (from blue to red) is related to the Torque value and where the bubble diameter is related to the Power value.
The green labeled designs belong to the Pareto frontier. The idea behind this visualization is to seek for big red bubbles, that are, by analogy, high Power and high Torque configurations. All these designs, as represented in the chart, are obviously the best solutions (the green labeled ones).

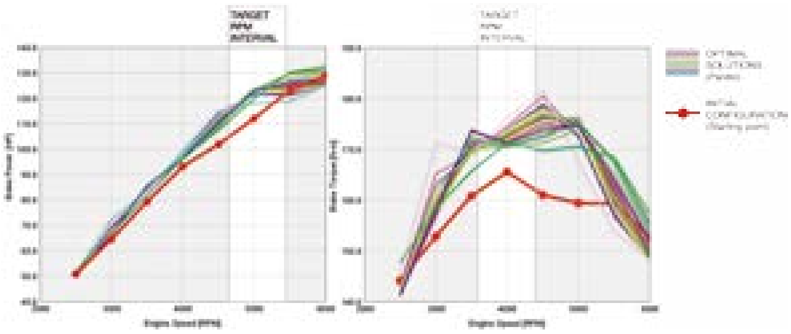


Fig. 7 - Power and Torque curves. Comparison of original manifold (RED) and optimal solutions (Colored) within target speed intervals

The GT-Power results are visualized in Figure 7. The plots show the functioning curves of the engines: the red curve is the initial configuration of the air intake manifold while the colored curves represent the Pareto design coming from the optimization. On the left plot, the brake power as a function of the engine speed is visualized, while the right plot points out the brake torque as function of speed. We can now understand that at 5000 rpm, the power curve of the engines with Pareto configurations is above the curve of the engine with the initial configuration. The same applies to the torque curve, where at 4000 rpm all the optimal solutions improve the original design.

The outcome of the first optimization phase is more aimed at establishing a methodology rather than at getting the optimization results themselves. This methodology might be scaled for more sophisticated manifold models. The second task is to apply the same procedure to an engine model with a more detailed parameterization of the geometry.

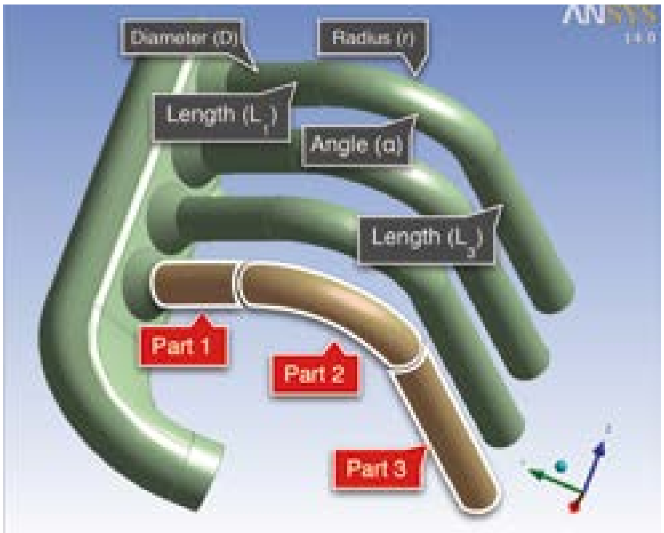


Fig. 8 - New Manifold Parameterization

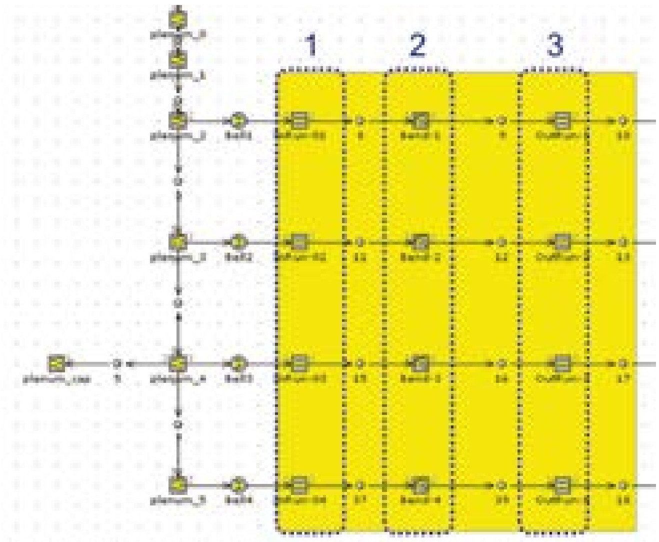


Fig. 9 - new GT-Power model developed taking into account the new geometry parameterization

Instead of using only three runner parameters for the whole manifold, in this step, the runners will be split into three parts (please refer to Figure 8):

1. Geometry Part 1: straight duct next to the inlet described by a diameter (D) and a length (L1).
2. Geometry Part 2: bend duct described by a radius (r) and an angle (α).
3. Geometry Part 3: straight duct next to the outlet described by a length (L3) as a function of other entities given by the

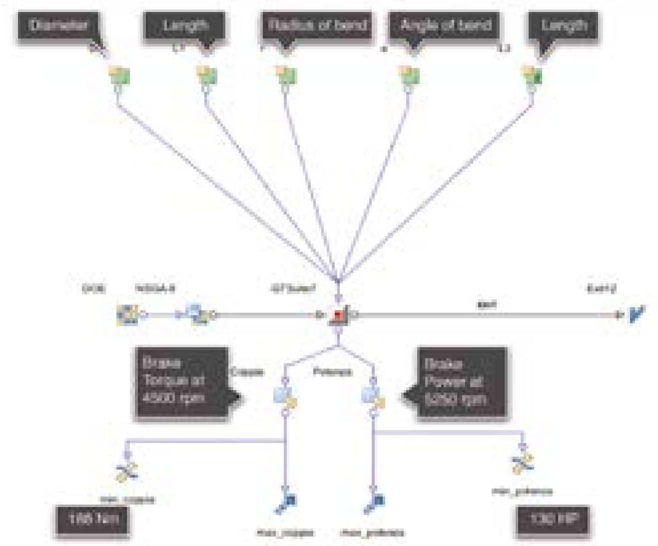


Fig. 10 - modeFRONTIER workflow adopting the new geometry parameterization with five runner parameters

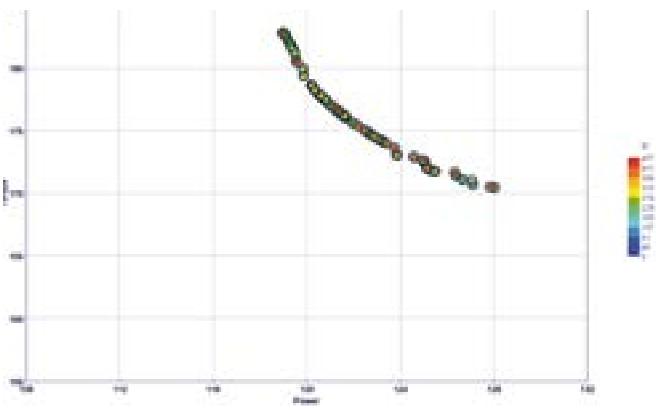


Fig. 11 - Targets bubble chart with brake Power (x-axis) and brake Torque (y-axis) displaying the results of the second optimization loop (new parameterization)

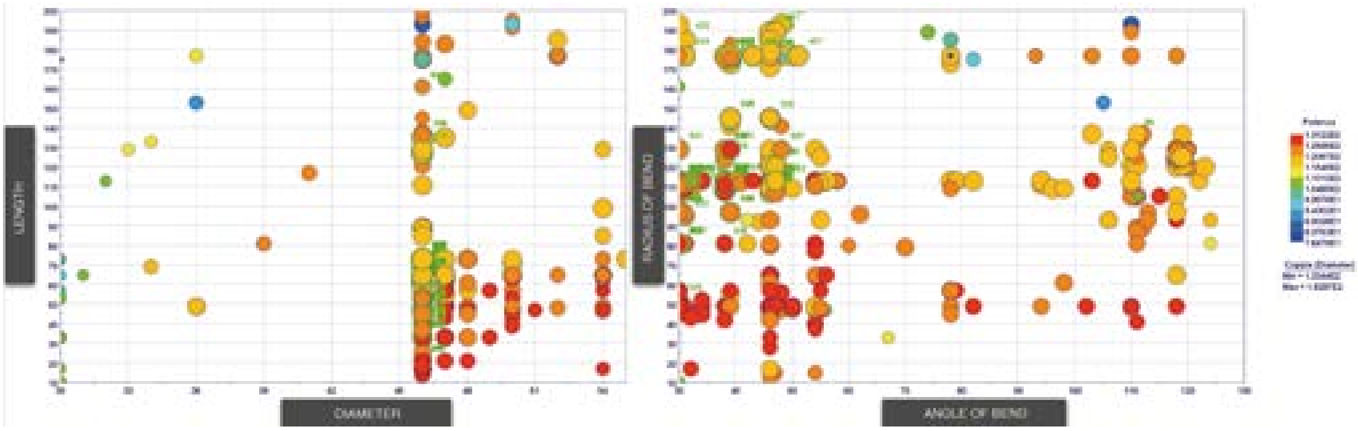


Fig. 12 - 4 dimension scatter chart with runner's Part 1 diameter and Part 2 angle of bend (xaxis), runner's Part 1 length and Part 2 radius of bend (y-axis), brake Power as color, and brake Torque bubble diameter. Green labeled are the Pareto designs

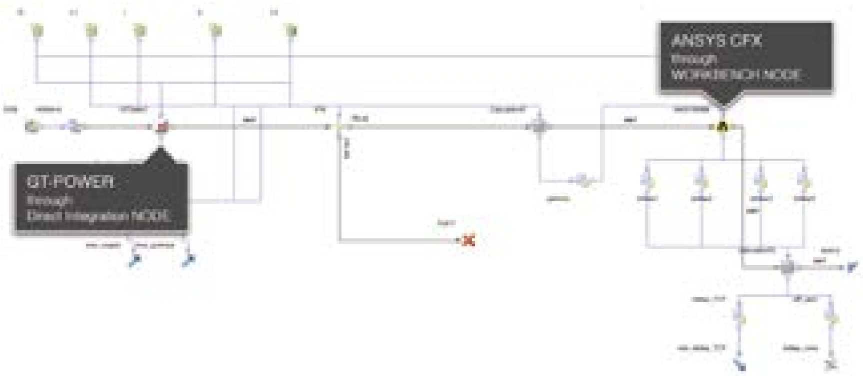


Fig. 13 - modeFRONTIER workflow adopting the new geometry parameterization with five runner parameters and including CFD analysis through ANSYS Workbench Node

geometric constraints. For example, the exit out of the plenum and the admission to the cylinders are fixed, hence the runners have to vary between these two points.

Since the geometry of the manifold is now different to the first 1D model, it is necessary to develop a novel GT-Power model (see Figure 9).
A new modeFRONTIER workflow must be created, a shown in Figure 10, with the new project diagram.
As a consequence of the new model, we obtain a new Pareto set on the Power vs Torque plot (Figure 11).

As far as the values of the input variables are concerned, we can see in Figure 12 in the second optimization loop, all the parameters plotted versus the design targets.
The results of the second phase assured the designers of the reliability of this design optimization procedure.
However, during the pre-design phase, it is also important to check the quality of the 1D analysis with a 3D CFD (Computational Fluid Dynamic) model. In addition, a CFD analysis will provide relevant information for improving the fluid-dynamic response of the manifold.

Moreover, the information that GT-Power can provide on the outlet region of the flow is not as reliable as the results from a 3D CFD analysis and simulation of the pressure drop and flow uniformity. For these reasons, CFX as CFD solvers have been included in the process flow of the modeFRONTIER multi-disciplinary environment. In Figure 13, we can discern that the CFX node has been incorporated in the modeFRONTIER workflow of the second optimization phase (Figure 10).

More specifically, since the runner should also satisfy the rigid geometry boundary conditions at the outlet, the manifold geometry is modeled using the ANSYS Design Modeler. The outcome of the use of a CFD tool in the optimization loop though has not yet been completely investigated, it is still in a work-in-progress phase.

Conclusion

The procedure adopted for optimizing the shape of a manifold by means of modeFRONTIER demonstrates its reliability and scalability.

As demonstrated during all the phases of the project, the adoption of a properly conceived design framework (input variables, outputs, objectives...) allowed the designer to keep the focus on the most important aspects of the design like modeling the right geometry and testing more solvers (from GT-Power 1D to ANSYS CFX 3D) while satisfying the customer's specifications and the geometry constraints.

Ing. Assunta Teducci - Magneti Marelli,
Ing. Francesco Franchini - EnginSoft

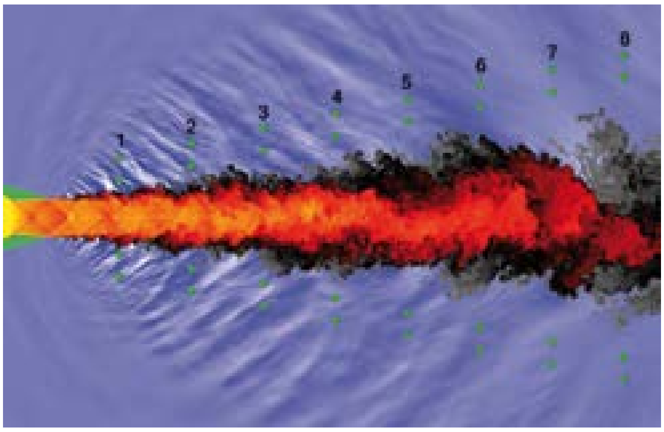
Assunta Teducci works in Magneti Marelli as CAE Engineer since 5 years dealing with structural simulation and fluid dynamics 1D Simulation (GT Power) of engine intake system, providing support to the new engine design.

For more information:
Francesco Franchini - EnginSoft
info@enginsoft.it

Million-core Supercomputer Barrier broken by Stanford Researchers

Stanford Engineering's Center for Turbulence Research (CTR) has set a new record in computational science by successfully using a supercomputer with more than one million computing cores to solve a complex fluid dynamics problem for the prediction of noise generated by a supersonic jet engine. The calculation was performed with Charles, the High Fidelity CFD code from Cascade Technologies Inc., the EnginSoft Partner in California, USA.

A new milestone in high performance computing was reached late Tuesday January 22nd 2013 evening when Stanford researcher and Cascade Technologies consultant Dr. Joe Nichols ran the CharLES solver on more than 1 million processor cores. This breakthrough happened during "Early Science" testing of the newly installed Sequoia supercomputer at the Lawrence Livermore National Laboratories (LLNL). The Sequoia IBM Bluegene/Q system is currently ranked No. 2 on the list of the world's most powerful supercomputers boasting 1,572,864 compute cores and 1.6 petabytes of memory connected together with a high-speed five-dimensional torus interconnect.



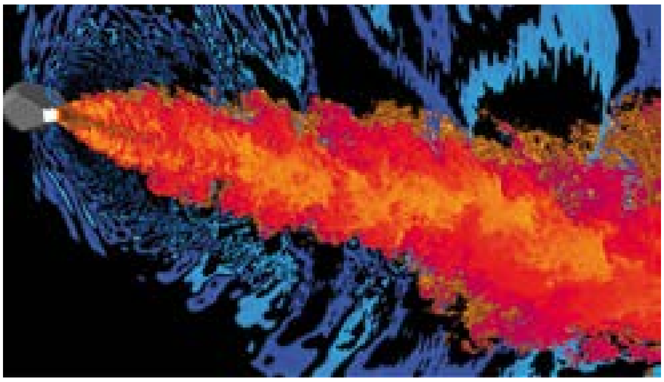
A snapshot of pressure and temperature from the "crackling" jet run during the scaling study.

THE PHYSICS OF NOISE

The exhausts of high-performance aircraft at takeoff and landing are among the most powerful human-made sources of noise. For ground crews, even for those wearing the most advanced hearing protection available, this creates an acoustically hazardous environment. To the communities surrounding airports, such noise is a major annoyance and a drag on property values. Understandably, engineers are keen to design new and better aircraft engines that are quieter than their predecessors. New nozzle shapes, for instance, can reduce jet noise at its source, resulting in quieter aircraft. Predictive simulations—advanced computer models—aid in such designs. These complex simulations allow scientists to peer inside and measure processes occurring within the harsh exhaust environment that is otherwise inaccessible to experimental equipment. The data gleaned from these simulations are driving computation-based scientific discovery as researchers uncover the physics of noise. These kinds of simulations can therefore help researchers discover how to build quieter aircraft engines. Under the direction of Professors Parviz Moin and Sanjiva Lele, the Stanford team has been working with the NASA Glenn Research Center in Ohio and the NAVAIR branch of the U.S. Navy to predict how loud an experimental engine will be without having to actually construct a prototype. Nichols explains that the acoustic energy of an engine is less than one percent of its total energy. Calculations have to be extremely precise in order to accurately model the noise an engine will generate. The technique is known as predictive modeling and it is an exacting process. The noise that a jet engine produces constitutes less than one percent of the device's total energy output, which means that accurately reproducing them in Computational fluid dynamics (CFD) simulations requires incredibly precise calculations.

CHARLES

CharLES™, Cascade Technologies flagship product, is a high-fidelity unstructured compressible flow solver for Large Eddy Simulation (LES), ideally suited for aeroacoustic applications involving high-speed flows and complex geometries. The numerical schemes embedded in CharLES provide a high-fidelity representation of turbulent fluctuations without introducing any artificial damping, i.e. numerical dissipation. Substantial effort is devoted to the efficiency of the algorithms and their applicability to large-scale parallel computations (on the order of



An image from the jet noise simulation. A new design for an engine nozzle is shown in gray at left. Exhaust temperatures are in red/orange. The sound field is blue/cyan. Chevrons along the nozzle rim enhance turbulent mixing to reduce noise. (Illustration: Courtesy of the Center for Turbulence Research, Stanford University)

150,000+ cores). All the physical models in CharLES were modified and enhanced to take advantage of the wealth of information provided by the direct representation of large-scale turbulent motions. Along with aircraft noise simulations, CharLES is being used to spruce up other areas of aircraft, such as to simulate turbulent flow over a wing, and propulsion systems used in hypersonic flight.

THE SUPERCOMPUTER

The 3,000 square-foot Sequoia IBM Bluegene/Q supercomputer is located at Lawrence Livermore National Laboratory (LLNL) in California (USA) and is among the most powerful parallel computing systems on the planet. It sports over 1.5million embedded processors 1.6 PB of memory and crunches numbers at a staggering 16.32 PFLOPS. The Sequoia's cores are arranged in a 5D Torus design wherein each core is directly connected to ten others. This greatly reduces latency even with cores two and three connections away. Read/Write functions are handled by these processors as well—some of which tap directly to the system's primary input/output channel through an 11th connection. While all 1.5 million cores may be necessary to calculate the nuclear weapons simulations that it is normally charged with, Joseph Nichols' research team from Stanford Engineering's Center for Turbulence Research harnessed just over a million of them for the jet engine research.

MORE CORES, MORE CHALLENGES

Supercomputers like Sequoia work by breaking down very large problems into smaller problems and distributing them across many machines and many processor cores. Typically, adding more cores makes the calculations faster, but it also adds complexity. At a certain point, calculations can actually become slower due to bottlenecks introduced by the communications between processors. A CFD simulation tasks all parts of a supercomputer because waves propagating throughout the tightly coupled simulation require a well-orchestrated balance between computation, memory, communication, and I/O. At the one million-core level, previously innocuous parts of simulation codes may suddenly become bottlenecks, and massive parallelism through all aspects of the software architecture is critical. Joe and other researchers from Stanford's PSAAP program and LLNL computing staff have been working closely together for a few weeks now to prepare for this unprecedented opportunity. So, together, they were glued to their terminals (more than usual) Tuesday (January 22nd 2013) afternoon and into the evening during the first "full-system scaling" window of the early science testing period to see whether initial extreme-scale science runs would achieve stable run-time performance after starting up. As the first CFD simulation passed successfully through its initialization phase, all were thrilled as they saw the code performance continue to scale all the way to and beyond one million cores. This means that the time-to-solution continued to reduce, enabling more science to be done with ever-faster turnaround times. These early science runs represent at least an order-of-magnitude increase in computational power over the largest runs performed using CharLES to date, enabling unprecedented fidelity and dramatically reducing time to solution. "These runs represent at least an order-of-magnitude increase in computational power over the largest simulations performed at the Center for Turbulence Research previously," said Nichols "The implications for predictive science are mind-boggling." The team hopes the results will help create quieter jet engines. Thanks to the Sequoia, Nichols thinks their research could go beyond just modeling into prescriptive design — in other words, figuring out what the optimum design would be. "It gave pause to a lot of people," Nichols said. "We were like: 'Whoa we can actually do that.'"

For more information:
Alberto Bassanese - EnginSoft
info@enginsoft.it

"Computational fluid dynamics (CFD) simulations, like the one Nichols solved, are incredibly complex. Only recently, with the advent of massive supercomputers boasting hundreds of thousands of computing cores, have engineers been able to model jet engines and the noise they produce with accuracy and speed," said Parviz Moin, the Franklin M. and Caroline P. Johnson Professor in the School of Engineering and Director of the CTR told Wired.

Prof. Parviz Moin

Analisi delle DIFETTOLOGIE NEL PROCESSO DI PRESSOCOLATA: contributo della simulazione numerica

Le attuali condizioni di mercato impongono alle realtà produttive tempi di risposta compressi, getti con elevate caratteristiche qualitative e prezzi competitivi. Per ottenere tali risultati l'attività di progettazione riveste un ruolo di estrema importanza risultando però onerosa in quanto soggetta ad una attività di verifica della qualità che prevede il passaggio obbligato attraverso una serie di campionature sul primo stampo per validare il processo di produzione per poi quindi avviarlo.
A causa delle specifiche condizioni di un processo, come quello di presso colata, non è semplice definire le cause che generano un micro o un macro difetto. Spesso la formazione di un giunto freddo, di un mancato riempimento o di un inestetismo superficiale critico possono derivare da una serie di concause di processo che se identificate permettono di individuare gli interventi correttivi. Il lavoro del progettista, che si avvale di uno strumento di simulazione virtuale, comprende la conoscenza del processo, la sua espressione "numerica" e la capacità interpretativa dei risultati forniti dallo strumento predittivo individuando quanto avverrà realmente in produzione, prevedendone l'insorgenza dei difetti e motivandone le cause. In fase di campionatura e produzione, la simulazione diventa strumento di indagine per identificare i parametri più opportuni per minimizzare i difetti realmente riscontrati. Lo strumento di indagine virtuale (simulatore) deve a sua volta garantire l'affidabilità dei risultati che fornisce rispecchiando quanto avviene nella realtà produttiva.

Il legame tra il campo virtuale e quello reale è fornito dall'osservazione scientifica di un fenomeno che, attraverso una analisi accurata delle sue caratteristiche, delle circostanze che lo producono e dei fattori che lo influenzano, permette la formulazione di valide teorie, in grado di riportare la spiegazione del maggior numero possibile di fatti sperimentali a un determinato numero di principi, attraverso una elaborazione matematica, di interpretare e, in certe condizioni, di prevedere l'evoluzione dei più svariati fenomeni.

Il calcolo numerico applicato alla fonderia

Da un punto di vista teorico, i fenomeni che stanno alla base dei processi di fonderia sono certamente conosciuti:

- il riempimento di uno stampo o di una forma per mezzo di una lega allo stato liquido segue le leggi della fluidodinamica (equazioni di Navier-Stokes); Si tratta, più precisamente, delle equazioni di continuita', di conservazione della quantità di moto, e dell'energia, alle quali si impone di soddisfare i bilanci di massa, quantita' di moto, ed entalpia. Esse possono essere scritte nella forma generale seguente:

  $$\frac{\partial}{\partial t}(\rho\Phi)+\frac{\partial}{\partial x_i}(\rho\Phi u_i)=\frac{\partial}{\partial x_i}\left(\Gamma_\Phi\frac{\partial\Phi}{\partial x_i}\right)+S_\Phi$$

  nella quale i termini rappresentano rispettivamente la non stazionarietà, la convezione, la diffusione ed un'opportuna sorgente. La variabile dipendente può indicare diverse quantità, come la frazione di massa, l'entalpia, la temperatura, una componente di velocità, l'energia cinetica di turbolenza. A seconda della variabile dipendente scelta, deve essere attribuito un significato diverso al coefficiente di diffusione ed il termine relativo alla sorgente. Ad esempio, se si particolarizza l'equazione (1) a rappresentare la conservazione della quantità di moto, la variabile dipendente diviene una componente della velocità, il coefficiente di diffusione diviene la conduttività termica, e la sorgente è data dalle forze esterne agenti.
- la solidificazione e il successivo raffreddamento della lega avviene secondo quanto previsto dall'equazione del calore di Fourier; Per il flusso termico qi, di un materiale caratterizzato da conducibilità termica K e rispetto ad un campo di temperature T, l'equazione si scrive:

  $$q_i=-K\frac{\partial T}{\partial x_i}$$

- le eventuali trasformazioni di fase cui può andare incontro la lega solidificata sono descritte in termini termodinamici e cinetici dalle leggi della metallurgia fisica.

Nella formulazione classica le equazioni di Navier-Stokes riguardano il moto laminare ma possono essere applicate a quello turbolento se a tutte le variabili vengono attribuiti valori istantanei. Risulta, pertanto, conveniente scomporre il valore istantaneo di φ (che in questo caso rappresenta le tre componenti del vettore velocità e la pressione) in una componente media ed una componente fluttuante.
In seguito a tale scomposizione viene applicata un'operazione di media all'equazione vettoriale della quantità di moto ottenendo in questo modo le equazioni "Reynolds avareged Navier-Stokes" (RANS): tali equazioni confinano il contributo dei termini fluttuanti (che rendono le simulazioni di problemi complessi non praticamente realizzabili a causa delle attuali limitate risorse di calcolo)

all'interno del tensore degli sforzi di Reynolds. Tale tensore introduce sei incognite aggiuntive (tensore simmetrico) al sistema di equazioni differenziali date dal prodotto delle componenti del vettore della velocità fluttuante. La modellizzazione di tale tensore, e di conseguenza le chiusura del sistema di PDEs (sistema di equazioni di bilancio) è affidata al modello di chiusura della turbolenza scelto.
Le nuove variabili di campo del modello di turbolenza sono l'energia cinetica turbolenta ed il coefficiente di dissipazione dell'energia cinetica turbolenta. L'insieme di equazioni di tale modello viene scritto per grandezze medie ed utilizzando parametri adimensionali. Inoltre si utilizza un algoritmo di calcolo in grado di simulare il comportamento pseudoplastico di un fluido non newtoniano.
Inoltre introducendo il coefficiente di tensione superficiale di uno specifico fluido è possibile tener in considerazione l'effetto di questa in funzione del fluido indagato e delle condizioni a contorno presenti (temperatura, velocità, pressioni e materiali a contatto).
Le equazioni di riferimento (1) e (2) sono trattate negli approcci di tipo numerico riconducendole in modo vario a sistemi di equazioni algebriche. I percorsi seguiti sono necessariamente diversi in relazione al metodo scelto, ma anche, all'interno di ciascun metodo, in relazione alle approssimazioni numeriche introdotte ed alle grandezze esplicitate.
Gli approcci numerici, seguiti dalla maggior parte dei software commerciali che trattano il problema della simulazione di processo in fonderia in modo completo, includono sia della fase di riempimento della cavità che la fase di solidificazione, e sono principalmente tre:

- il metodo delle differenze finite (FD): metodo per risolvere numericamente equazioni differenziali, prevalentemente ordinarie anche se sono spesso usate come schema di avanzamento nel tempo per problemi alle derivate parziali;
- il metodo dei volumi di controllo (CV): metodo utile nell'integrazione di equazioni differenziali alle derivate parziali. Tali equazioni vanno integrate in un volume sui cui confini sono imposte le condizioni al contorno;
- il metodo degli elementi finiti (FE): metodo numerico atto a cercare soluzioni approssimate di problemi descritti da equazioni differenziali alle derivate parziali riducendo queste ultime ad un sistema di equazioni algebriche.

A tali richieste per i codici di calcolo fluidodinamico la miglior risposta viene fornita dal metodo dei volumi di controllo che permette di descrivere il flusso, in questo caso della lega, con buona precisione tanto più spinta quanto maggiore è il numero dei volumi che discretizzano la geometria oggetto di studio. Il metodo inoltre si presta ad una più semplice ed immediata formulazione della discretizzazione stessa rendendo il sistema di meshatura più agile e semplice nell'impostazione da parte dell'utilizzatore.

Affidabilità dell'analisi virtuale

Nell'utilizzo di qualsiasi codice di calcolo, la bontà dei risultati finali è diretta conseguenza della correttezza delle informazioni fornite in ingresso. In linea di principio, si può dunque affermare che le fasi principali di un processo di fonderia possono essere descritte mediante formulazioni matematiche, più o meno complesse, descritte precedentemente. L'affidabilità dell'analisi fisico-matematica dipende, a questo punto, in maniera significativa quindi dalla correttezza con cui vengono definite le condizioni al contorno e dalla conoscenza delle proprietà dei materiali in esame.
Mediante le condizioni al contorno si possono prendere in considerazione parametri operativi essenziali del processo quali:

- la configurazione geometrica del sistema che si vuole simulare (colata, stampi, circuiti di termoregolazione);
- i parametri che caratterizzano il processo: la temperatura di colata, i flussi di metallo fuso, le varie fasi operative che scandiscono il tempo ciclo (apertura delle parti stampanti, istanti di lubrifica, chiusura delle parti stampanti, iniezione della lega nello stampo, fase di solidificazione).

La precisione di tali informazioni risulta fondamentale tanto quanto i metodi di calcolo numerico applicati e citati precedentemente. Maggiore quindi è la attinenza e precisione delle informazioni immesse nel codice di calcolo maggiore sarà la precisione del risultato virtuale.
Ulteriore input fondamentale nel calcolo virtuale consiste nella caratterizzazione numerica delle proprietà termo fisiche dei materiali (leghe e acciai per le parti stampanti), devono essere note infatti (o ragionevolmente stimate):

- per la lega: la temperatura di liquidus e di solidus, e, in funzione della temperatura, la viscosità, la densità, la conducibilità, la diffusività termiche e tensione superficiale;
- per lo stampo (e per le anime): le proprietà termiche e le caratteristiche di eventuali sistemi di raffreddamento;
- i coefficienti di scambio termico che regolano il passaggio del flusso di calore attraverso differenti corpi a contatto in funzione delle condizioni di contatto presenti (rugosità superficiale, pressione di contatto).

Tali informazioni possono essere ricavate sperimentalmente con test di laboratorio specifici affiancati all'utilizzo di software che ne permettono la loro identificazione. Chiaramente la competenza e le conoscenze specifiche nel campo metallurgico completano il quadro della caratterizzazione specifica di un materiale.

I risultati di una simulazione di pressocolata

Pur essendo un processo altamente automatizzato, la pressocolata denota una molteplicità e variabilità di parametri che non consentono la semplice individuazione delle cause che generano un micro o un macro difetto. Ad esempio, la formazione di un giunto freddo o di un inestetismo superficiale critico, sino al mancato riempimento, possono derivare da una serie di concause di processo che, se identificate, permettono di individuare gli interventi correttivi.
Per una produzione di massa, risulta fondamentale la ricerca della soluzione ottimale sia in termini qualitativi che di efficienza produttiva: la riduzione di pochi secondi del tempo ciclo si ripercuote positivamente sul numero di pezzi prodotti giornalmente; la minimizzazione della quantità di lega di iniettare per il singolo pezzo abbatte i costi di materia prima, di energia e riciclo; pochi punti percentuale nella riduzione dello scarto si può tradurre in

interessanti vantaggi economici; infine il prolungamento della vita delle attrezzature riduce l'incidenza sul costo del singolo pezzo. Le tecniche di simulazione virtuale supportano adeguatamente l'identificazione delle difettologie, derivanti da un processo produttivo sul prodotto finale, e le loro cause.
È proprio questa l'essenza della simulazione numerica: visualizzare la storia del processo, mediante le grandezze più opportune, in modo da comprenderne con un notevole grado di accuratezza l'evoluzione. In sostanza il processo reale di fonderia e il processo "virtuale" simulato al calcolatore compiono un percorso "parallelo": la simulazione, se correttamente impostata, è in grado di descrivere la caratteristiche finali di un getto ancor prima che questo venga colato.
I risultati che si possono ottenere da un sistema di analisi virtuale possono essere suddivisi nelle seguenti categorie:

- rappresentazioni del campo fluidodinamico del sistema, cioè delle distribuzioni di massa e di temperatura durante il riempimento della cavità;
- rappresentazioni del campo termico del getto, cioè delle distribuzioni di temperatura durante la solidificazione e il raffreddamento;
- rielaborazioni delle informazioni derivanti dal campo fluidodinamico e termico, tali da consentire una valutazione della eventuale difettosità del getto;
- visualizzazione delle distribuzioni di temperatura nello stampo nel corso di tutte le fasi del processo produttivo;
- visualizzazione dell'evoluzione tensioni e deformazioni residue del getto e delle sollecitazioni sulle parti stampanti.

L'elevata conoscenza dei difetti e l'opportuna interpretazione dei risultati virtuali è un prerequisito fondamentale all'uso della simulazione.
Di seguito vengono riportati alcuni brevi esempi di analisi virtuali poste a confronto con la produzione reale.
L'identificazione della presenza di inglobamenti d'aria, che si manifestano in bolle più o meno grosse all'interno o superficialmente al pezzo, sono identificabili attraverso l'utilizzo del criterio "Airpressure". Grazie all'attivazione dell'opzione raggi X virtuale è possibile identificare le zone affette dalla problematica ed il livello di criticità (Fig. 1). L'analisi viene estesa valutando la dinamica di riempimento per individuare le cause del mancato sfiato d'aria verso gli appositi pozzetti.

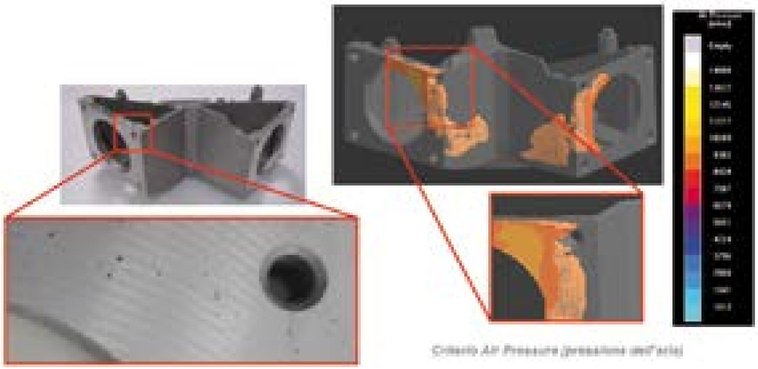


Fig. 1 - Porosità dovuta ad inglobamento d'aria a fine riempimento

L'identificazione di fronti di flusso che tendono a sottoraffreddare durante la fase di riempimento permette di prevedere la problematica legata alla formazione di giunti freddi o addirittura mancato riempimento (Fig. 2) sfruttando l'analisi della dinamica di riempimento in funzione della distribuzione della temperatura della lega.

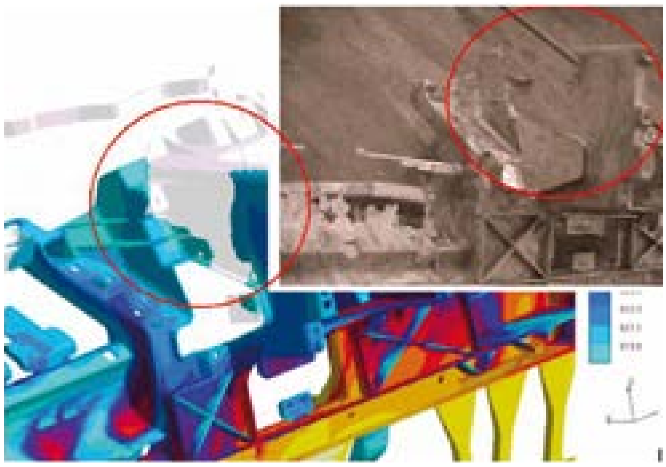


Fig. 2 - Mancato riempimento del getto dovuto ad un eccessivo abbassamento del fronte di flusso

L'analisi della fase di solidificazione permette di identificarne la sua evoluzione visualizzando l'isolamento delle zone massive che danno origine alla formazione di porosità da ritiro (Fig. 3). L'applicazione dell'opzione raggi X virtuale, che permette di rendere trasparente le parti già solidificate mettendo in evidenza le zone critiche, fornisce all'utente un valido supporto permettendo di identificare con maggior facilità le difettologie presenti.

Come detto precedentemente l'analisi virtuale permette di focalizzare l'attenzione anche sulle criticità inerenti alle parti stampanti. Una analisi svolta a regime termico, considerando cioè più cicli produttivi, fornisce un valido contributo per identificare per esempio quali siano le zone dello stampo soggette a deterioramento dovuto allo shock termico al quale sono sottoposte, predicendo quindi la vita a fatica dello stampo stesso (Fig. 4).

La nuova frontiera della simulazione: l'ottimizzazione multi-obiettivo

Và detto che gli attuali codici di calcolo non devono essere considerati come dei sistemi esperti, soprattutto in fase di progettazione, non è certo possibile rimpiazzare automaticamente l'apporto e l'esperienza del progettista di fonderia. L'approccio al lavoro può essere di tipo "prova e correggi", ed ha lo scopo di Ottimizzare la produzione sia dal punto di vista geometrico sia di parametri di processo.
A causa del gran numero di variabili che lo definiscono, il processo di fonderia è sicuramente uno dei problemi più difficili da ottimizzare, in quanto non esiste in genere una sola soluzione che soddisfi al meglio gli obiettivi preposti.
Per poter scegliere la miglior configurazione, mix dei parametri che controllano il processo, è necessario confrontare un elevato numero di soluzioni potenzialmente possibili.

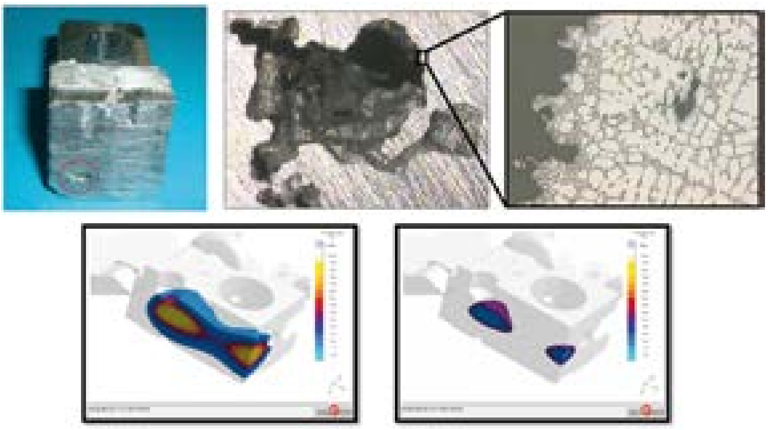


Fig. 3 - Identificazione di porosità da ritiro

Gli esempi applicativi hanno inoltre dimostrato l'efficacia della simulazione numerica applicata alla fonderia fornendo un valido contributo al progettista che prende completamente coscienza del suo elaborato ancor prima di effettuare la fase di campionatura. Il suo utilizzo, soprattutto nell'ambito del concetto di "simultaneous engineering", diventa veramente prezioso in termini di qualità finale del pressocolato e di gestione dei costi e delle risorse introducendo il concetto di campionatura virtuale che permette di ridurre drasticamente il numero delle campionature reali necessarie al benestare per l'avvio della produzione. Ulteriore vantaggio fornito dall'utilizzo dell'analisi virtuale riguarda la possibilità di esplorare differenti e spesso nuove configurazioni di colata incrementando il Know-How del progettista e aziendale, fornendo nuovi spunti per migliorare l'attività produttiva dal punto di vista della qualità del getto e degli sprechi (si pensi per esempio alla possibilità di poter ridurre le dimensioni del sistema di colata o dei pozzetti riducendo così la quantità di lega necessaria per produrre la singola stampata).

L'approccio virtuale trova infine la massima espressione con l'applicazione dell'Ottimizzazione Automatica che permette la gestione più accurata e precisa delle innumerevoli variabili garantendo un ulteriore incremento delle prestazioni del processo produttivo.

È possibile in questo caso sfruttare sistemi di Ottimizzazione Automatica Multi Obiettivo che coordinano le simulazioni di pressocolata al fine di identificare la configurazione ottimale che minimizzi/elimini i difetti incrementando al massimo la qualità del prodotto.

Il confronto di un elevato numero di soluzioni potenzialmente applicabili è reso possibile dall'introduzione di variabili che possono gestire la forma geometrica e/o i parametri di processo all'interno di determinate condizioni imposte dal progettista, il quale definirà inoltre gli obiettivi a cui tendere per individuare la configurazione ottimale. Tali obiettivi, molteplici e contrastanti, possono riguardare gli aspetti relativi alla qualità del getto e/o alla vita delle parti stampanti portando così a definire la configurazione ottimale per il processo produttivo (Fig. 5).

Chiaramente l'incremento del numero delle variabili utilizzate e degli Obiettivi richiesti determina la complessità del problema con l'inevitabile difficoltà da parte del progettista di individuare la miglior configurazione possibile. Lo strumento di Ottimizzazione permette quindi di controllare con la massima precisione quale sia l'effetto delle modifiche delle variazione delle variabili sui risultati attesi per definire il progetto in tutti i suoi particolari.

Conclusioni

La complessità del processo di presso colata pone il progettista in una condizione di estrema difficoltà nel ricercare la soluzione sfruttando la sola esperienza. Per garantire la qualità di un prodotto in tempi rapidi contenendo i costi, diventa necessario avvalersi di strumenti di indagine virtuali che aiutino a prevedere le problematiche di produzione ancor prima di mettere in opera il progetto. Lo strumento di simulazione deve a sua volta essere robusto ed affidabile.

Quanto esposto ha messo in evidenza quali siano le caratteristiche, in termini matematici, necessarie per un software di simulazione, ma come sia altrettanto importante la precisione delle informazioni da introdurre nel codice di calcolo per avere dei risultati utili e di supporto alla fase di progettazione.

Per maggiori informazioni:
Giampietro Scarpa - EnginSoft
info@enginsoft.it

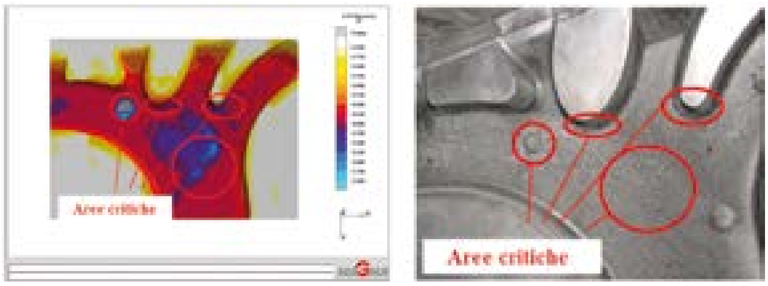


Fig. 4: Analisi della vita a fatica di uno stampo

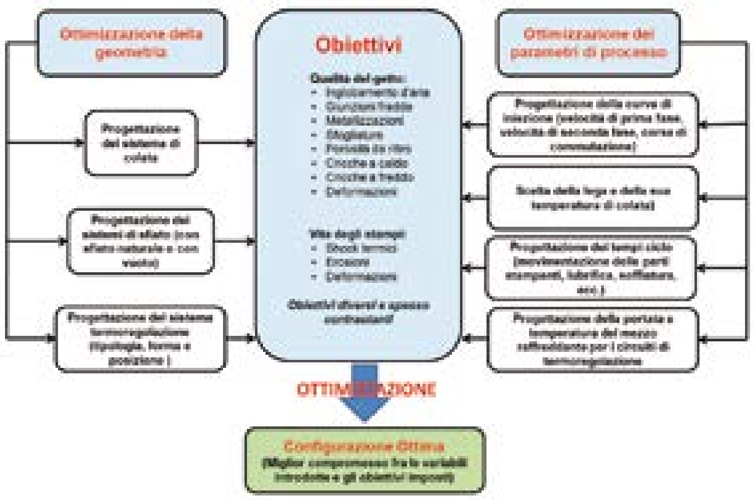


Fig. 5: Ottimizzazione Automatica Multi Obiettivo: Schematizzazione dei differenti approcci

MECHATRONICS: Enabling technologies for increasing competitiveness

Manufacturing plays a key role in today's economy. It creates wealth and employment from direct industrial activities and linked supplier business.

This is so despite the fact that the manufacturing sector has been experiencing a decline for some time. From 1957 to 2007, the share of the manufacturing industries in the U.S. GDP declined from 27% to less than 12%, although much of this trend actually reflects declining prices for manufactured goods. In 1969, manufacturing accounted for 26% of national employment in the US, today only about 9% are dependent on this sector. The same trend can be witnessed in Italy: from 1976 to 2010, the value added from the manufacturing sectors moved from 29,6% to 16,6% and the employment from 28,1% to 17,1%. One reason for this is the increasing relevance of emerging countries. Figure 1 shows the rapid growth of developing countries from 2000–2007 across all sub-sectors of manufacturing opposed to developed nations.

Despite the fact that the share of manufacturing in employment and GDP has decreased in recent decades, the key sector has become even more important. Manufacturing is the most capital-intensive and productive sector of the economy and hence of great importance for the development of new technologies and their commercialization. Manufacturing has the largest employment and output multipliers due to the many indirect jobs created by the diverse production activities. When we consider that it is estimated that each job in manufacturing is linked with two additional jobs in high quality services, it becomes clear that manufacturing is and will be crucial for increasing the employment rate in Europe.

Europe is one of the main players in the global economy, and both its process and discrete manufacturing industries contribute to a large extent to this situation. Manufacturing, in fact, represents the largest portion of the NACE sections within the EU-27's non-financial business economy, both in terms of employment and value added. It accounted for 24.2 % of the European workforce in 2008 and 27.1 % of the generated value added. 2.1 million manufacturing enterprises employed 33.0 million people in Europe in 2008. Nearly 70 million were employed in service-related manufacturing jobs. Moreover, when we talk about research, the sector accounts for 66% of total private investments.

In the last decades, European manufacturing industries have experienced dramatic changes due to global trends and the aforementioned challenges. The growth of emerging countries shaped a new map of the world's manufacturing hubs, and mature economies are facing problems in maintaining their leadership in the global marketplace.

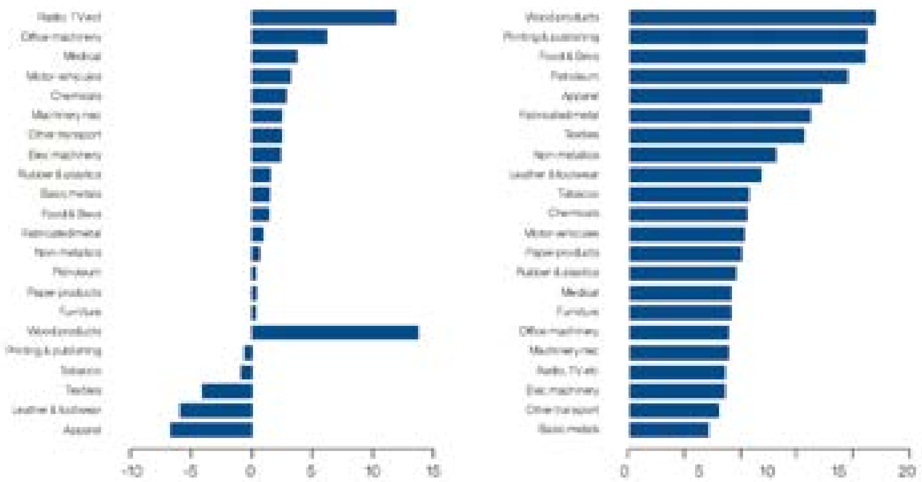


Fig. 1 - Growth of value added in developing countries (on the right) from 2000–2007 across all sub-sectors of manufacturing vs. developed nations (on the left)

High customization led to an important proliferation of product variants, with shorter planning horizons and product life cycles, and decreasing batch sizes. Similarly, the lowering of product prices requires an improved manufacturing approach based on leanness, flexibility, agility, and re-configurability, to remain competitive. The disintegration and globalization of the supply chain by new international markets, as well as the advances in Information and Communication Technologies, pushed companies to separate their production and business processes in intermediate steps, and to allocate them to countries and facilities that are able to achieve best performances.

Mechatronics: as enabling Technologies

Based on some analyses we performed in the past years, we would like to present and explain the important role of mechatronics further in this article. For this purpose, we also take into account the roadmap of the EFFRA – European Factories of the Future Association. According to this roadmap, the challenges that European companies are facing concern two main areas: Sustainability in manufacturing (which means economical, environmental and social sustainability) and the capability to manufacture the products of the future.

The first challenge requires to develop manufacturing processes which are sustainable; this comprises/means:

- Environmental sustainability is obviously a growing need. Crucial topics include: climate change, the depletion of resources, the uncertainty of access to resources, the effect of emissions on the environment and on public health, etc. These are all phenomena that require European manufacturing to become less dependent on energy and to use raw materials more efficiently, while reducing emissions and waste;
- Social sustainability: manufacturing can only generate welfare for citizens if manufacturing environments provide adequate workplaces to citizens.
- Economical sustainability is key to realizing any sustainability. Environmental and social sustainability can only be achieved if manufacturing is competitive and is able to generate the income required to pay knowledge workers and to invest in environmentally friendly and worker-friendly factories.

The second challenge concerns the manufacturing of the products of the future. In order to maintain long-term strategic competitiveness, Europe must assume a leading role in the design and production of products which are able to meet currently unsatisfied needs, and which have the potential to open new markets. In addition, societal challenges such as energy constraints, health and an aging society, stimulate the emergence of new products. Hence, developing the capability of manufacturing these products is a critical requirement for European welfare.

So, when we look at this scenario, at these factors, which is the role of mechatronics? To answer this question, let us first define the term mechatronics: "Mechatronics is the synergistic combination of precision mechanical engineering, electronic control and systems thinking in the design of products and manufacturing processes" (IFAC, the International Federation of Automatic Control). From this definition, it is evident that mechatronics is a combination of different sciences, from mechanical to electronics, which aims at improving the performances of products and manufacturing processes. From another point of view, it is possible to consider mechatronics as an enabling technology for the improvement of products and processes in the manufacturing context. Also the EFFRA Association defines mechatronics as enabling technologies to reach the targets of sustainability and innovation as described before. According to this approach, the following could be considered as the most relevant mechatronics technologies, which will play a major role in addressing manufacturing challenges:

- Control technologies: model-based development of machine control software, learning controllers, control methods that allow the adaptation of both feedback and feed forward control signals to changing environments, optimal control techniques, real-time communication technologies;
- Human Machine Interface;
- Monitoring and diagnostics: advanced signal processing or virtual sensing, model-based, signal-based, data mining-based methods for a wide range of condition monitoring applications, e.g. event pattern detection, diagnostics, anomaly detection, prognostics and predictive maintenance;
- Advanced structural system architectures: reconfigurable machine architectures;
- Energy technologies: energy storage components, energy harvesting technologies;
- Integrated product-process-production system design and simulation techniques.

A study conducted by the Intellimech Consortium within the MECH4APP district, a district of Mechatronics of the Lombardy Region built by 82 companies from diverse industrial fields, shows the relevance of these technologies from an industrial point of view. Figure 2 depicts the results of this analysis.

From the results of this study, it becomes evident how, from an industrial perspective, the most relevant technologies for the future will be those related to the control, the human machine interaction and to energy efficient techniques. Beyond that, the integrated product-process design approach that focuses on the use of simulation techniques, is regarded as one of the most relevant technologies in today's landscape.

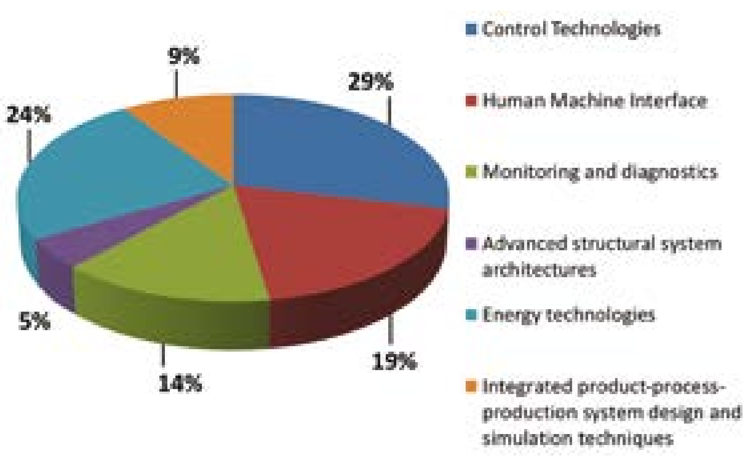


Fig. 2 - Analysis of the most relevant enabling technologies

A Case Study: Experience from Intellimech Consortium

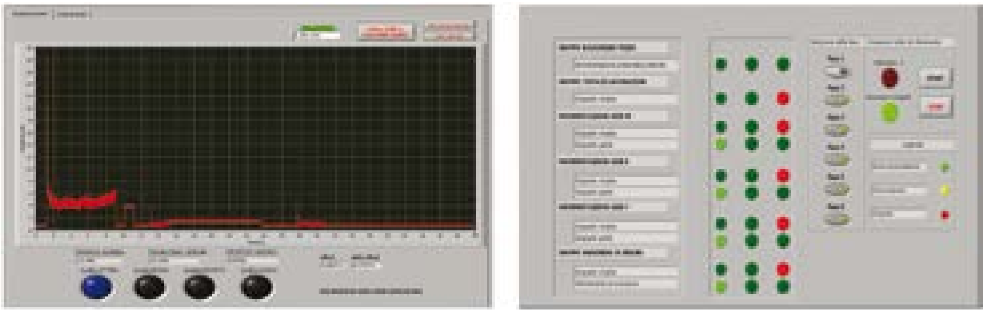


Figure 3: Application of ESA (Electrical Signature Analysis) for quality control of a vending machine (left) and as diagnostic tool for balancing a machine (right)

In 2007, the Intellimech Consortium, a private-led consortium of companies aiming at developing interdisciplinary research in the field of mechatronics, was founded. The consortium, which includes 16 companies operating in different areas, from braking systems to an assembly line manufacturer as well as experts for optimization and simulation processes, aims at sharing experiences and developing mechatronics technologies which can be applied in several fields. The diversity of companies is considered a value-added resource.
In fact, the basic idea and motivation is that the same technology can be applied for different purposes, in different industrial sectors, and sometimes also in the same industrial segment but for different purposes. This is a key element which justifies also the presence of direct competitors in the consortium; these companies actually share the investments for applying a mechatronics technology that was developed in different areas of product and/or production processes. The latter also suits very well the industrial scenario in Italy which is characterized by SMEs that need to share their efforts and investments to reduce the costs for R&D in order to stay competitive on the global market.

As an example, one of the projects carried out by Intellimech regards the electric signature analysis as a condition-based maintenance technique for the health assessment of industrial plants. ESA is the procedure of capturing the equipment's supply signals (current and/or voltage) and analyzing them to detect malfunctions (not only electrical ones) or incipient faults. The idea is that any variation in an electro-mechanical system generally produces correlated variations or disturbances in the current and voltage supply line of the electric motor powering the system. ESA can be used to analyze these small perturbations and to match them to their source. The resulting time and frequency signatures calculated with ESA reflect loads, stresses, and wear throughout the system and allow extracting an extensive range of mechanical diagnostic information.

This technique has been applied in two cases as shown in the figure 3 below:

- it has been used in the vending machine sector by two competitors. The technique has been developed jointly, but its application is different (one focuses on products while the other is applied to processes for quality control for instance);
- the same technique has been applied for balancing machine and diagnostic tools for assessing machine conditions.

It is evident how the same technique can be applied in different industrial scenarios (vending machine or machinery) and also in different areas within a company (e.g. for product improvement or quality control). How and where to apply the technology is a strategic decision which a company must take according to its culture and priorities, but the development of the technology can be carried out together in order to share and reduce the investments.

Conclusions

The role of manufacturing is rapidly changing in the modern markets. Even if, from a statistical point of view, its relevance is decreasing in the developed countries, it does represent a key factor for economic sustainability and growth in Europe in terms of wealth and employment. The increasing relevance of developing countries forces companies to face new challenges in the manufacturing context. The high customization of products and the increasing intelligence behind these products push companies to invest in manufacturing processes in order to achieve flexibility, re-configurability and agility for new products. Mechatronics is an enabling technology for reaching all these targets. Europe does recognize the important role of these technologies as enablers for increasing the competitiveness of companies.

In this context and as demonstrated by the activities carried out by the Intellimech Consortium, mechatronics represent a technology for sharing experiences and for associations of companies that can develop and share the same technology while reducing the investment cost per capita. This is very important for our industrial landscape as Italy sees a large presence of SMEs which should share more investments in R&D and innovation.

INTELLIMECH
CONSORZIO PER LA MECCATRONICA

Finally, it is worth mentioning the role of education. In Italy, education is also supported by SMEs which offer a high value added. The Intellimech Consortium sensitizes companies for these new industrial and market characteristics and highlights the opportunities involved in the same.

For more information:
Stefano Ierace - Intellimech
stefano.ierace@intellimech.it

ANSYS: Next Generation Product

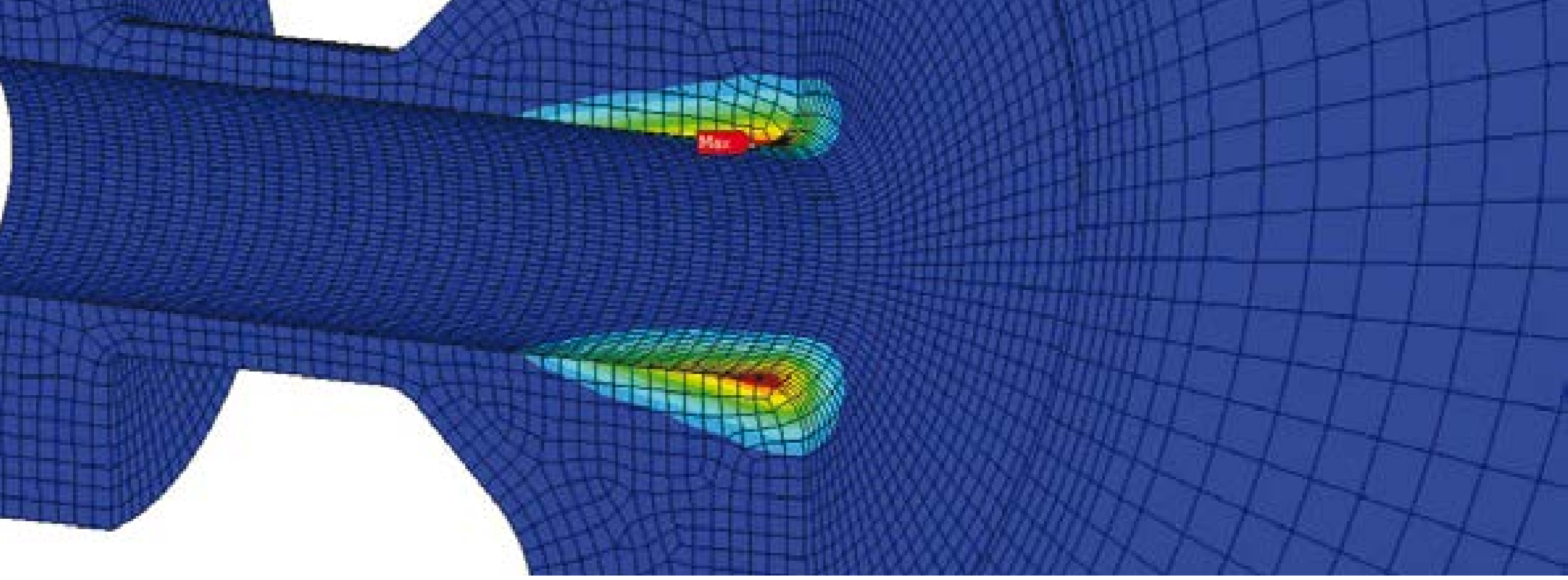
L'annuale appuntamento ANSYS ha avuto luogo anche quest'anno a Gennaio tra le innevate colline della Pennsylvania presso Faminghton a circa 2 ore di autobus da Pittsburg.
La sintesi del presidente Jim Cashmann e gli interventi dello staff dirigenziale hanno disegnato lo stato attuale della diffusione della tecnologia in termini di incremento sui singoli mercati mondiali nel 2012 ed inoltre sono state indicate alcune linee generali di sviluppo prodotto nelle tre declinazioni Ebu (Elettromagnetismo), Mbu (Meccanica), Fbu (Fluidodinamica) per il prossimo quinquennio.
Le conclusioni e i contenuti delle affermazioni sono state sicuramente positive nel senso di aver comunque affrontato e superato il periodo della crisi in quanto la crescita del mercato europeo, per esempio, si è attestato al 13%, il dato è particolarmente significativo relativamente alle arcinote difficoltà economiche della area nel passato anno; nei dettagli tale risultato si spiega con balzi positivi in paesi quali Turchia e Russia europea e dal successo di alcune tecnologie nuove come l'Hfss in Inghilterra; la "vecchia" Europa pur non ottenendo risultati sbalorditivi, ha confermato la solidità della tecnologia presente nelle maggiori aziende ad innovazione tecnologica del vecchio continente: Volkswagen è stata nel 2012 un grosso successo in un mercato, l'automobilistico, di cui giornalmente non abbiamo riscontri positivi sulle testate giornalistiche.

NEXT GENERATION PRODUCT, questa è la sintesi delle linee di sviluppo per le tre discipline del software; in altri termini parte un processo di rigenerazione dell'intero pacchetto che verrà rivisto al fine di aumentarne la produttività tecnica lavorando principalmente sugli aspetti multidisciplinari, sulla parametrizzazione e sulla capacità di analizzare modelli complessi costituiti da un elevato numero di elementi in tempi industrialmente congrui.
I target sono in meccanica statica 200 milioni di elementi, in non linearità 20 milioni di elementi in modale 50 milioni di elementi ed in transitorio 10 milioni di elementi per 1000 secondi; nel campo CFD 500 milioni di celle per una steady state e 100 milioni per un transitorio, ovviamente su postazioni hardware adeguate e spingendo al massimo su solutori paralleli; ovvio quindi gli sforzi a lavorare su tecnologia HPC come già da qualche anno si aveva evidenza.

La parte teorico documentale non è affatto trascurata ma anzi si avrà modo di aver un nuovo "customer portal" con 22000 pagine di documentazione in cui saranno illustrati esempi e training sintetici.
Ancora una volta è stato sottolineato l'importanza del "Channel Partner" (EnginSoft ha tale ruolo) sia per il ruolo squisitamente tecnico nel fornire assistenza agli utenti, sia per la esperienza che taluni hanno mostrato in alcuni settori; EnginSoft è stata in tal senso invitata ad illustrare sotto il profilo FEM alcune "storie di successo" in campo oil&gas, talune realizzate in collaborazione con ESSS Channel Partner brasiliano di ANSYS e privilegiato collaboratore di EnginSoft in alcune commesse internazionali.

I "council meeting" organizzati dai Channel partner sono fortemente apprezzati perché le informazioni circa le esigenze tecnologiche di sviluppo prodotto costituiscono materiale altamente significativo e congruo a impostare gli investimenti di ricerca sulle nuove "capabilities" del software così come le analisi sintetiche circa le richieste di assistenza del supporto.
In conclusione il futuro può essere senz'altro vivace in termini di nuove sfide su tematiche non standard, quali l'area compositi di cui EnginSoft è nel gruppo COE di ANSYS e relativamente al alcune commesse su progetti di ricerca condivisi con Alenia Aermacchi (Progetto Khira) sia sulla razionalizzazione del software all'intero di nuove e vecchie realtà industriali italiane con una rinnovata voglia di innovazione tramite per esempio l'applicazione di nuove tecnologie quali l'ACT (ANSYS Customization Tecnology).

Per maggiori informazioni:
Roberto Gonella - EnginSoft
info@enginsoft.it



EnginSoft implementa il modello di Creep "MCP OMEGA" in ANSYS

Nell'ambito della progettazione di tutti i componenti per l'industria Oil & Gas, Chimica e Nucleare e genericamente definiti "recipienti in pressione" i maggiori organi normatori (ASME, PED, API) hanno introdotto e sostengono criteri di verifica ed accettazione riferiti all'utilizzazione di modelli ad elementi finiti

Questi metodi di verifica, indicati come design "by analysis", integrano e completano le valutazione più snelle e semplificate che nelle fasi preliminari del progetto vengono operate mediante l'approccio "by formula" consentendo di affinare il progetto e rimuovendo eccessi di conservativismo.

La verifica di dettaglio, mediante simulazione, consente infatti una rappresentazione dettagliata e realistica del problema in tutti i suoi aspetti (dalla geometria, ai materiali alle condizioni al contorno termiche e meccaniche del problema) senza la necessità di doversi ricondurre ad analogie o semplificazioni.

A questa possibilità le normative di settore impongono una opportuna azione di regolamentazione prescrivendo linee guida sulle metodologie applicabili e valori precisi per le proprietà dei materiali impiegati e per i loro valori di resistenza ammissibile.

Un esempio in questa direzione è fornito, per esempio, dalla adeguata presa in conto del comportamento a creep di materiali sottoposti ad un regime di sollecitazione nel campo delle alte temperature: i codici ASME, principale riferimento normativo nel campo dei recipienti in pressione, prevedono che la valutazione sulla durata di vita dei componente e le eventuali azioni per la loro sostituzione o riparazione (FFS, Fitness for Service ref. API 579-1/ASME FFS-1) di apparecchiature realizzate con un ampia famiglia di acciai tengano in conto del Creep mediante l'utilizzo del modello "MPC Omega".

Lo stesso modello di creep è richiesto (attraverso il Code Case 2605) per validare la progettazione ad alta temperatura (tra i 370 °C e i 474 °C) di alcune leghe di acciaio basso legate (2.25 Cr – 1 Mo - 0.25 V): in questo caso vengono esplicitamente prescritte analisi inelastiche che contemplino contemporaneamente i problemi di plasticità, creep e danneggiamento a fatica.

Il modello MPC Omega discende dal metodo Omega per la valutazione della vita a fatica in regime di Creep sviluppato a fine anni 90 da M.Prager e poi successivamente migliorato e validato presso il Materials Properties Council (MPC), fondato dalla ASME stessa e da ASM International, ASTM e Engineering Foundation. Il metodo MPC Omega ha la peculiarità, rispetto a quelli utilizzati nella maggior parte delle simulazioni ad elementi finiti (es. Norton, Garofalo), di spingersi fino alla simulazione del fenomeno del creep terziario (Figure 1): in

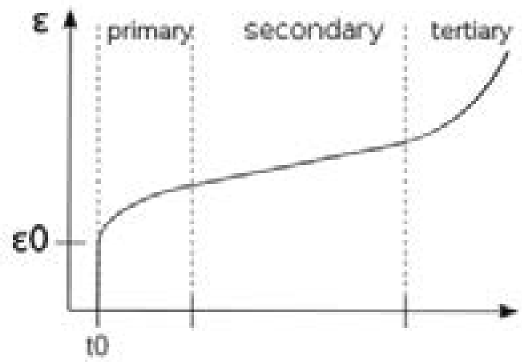


Fig. 1 - Le tre fasi del creep

questa fase il metallo comincia a danneggiarsi e lo strain rate aumenta velocemente fino al collasso o la rottura localizzata.

Il modello si basa sull'introduzione di un parametro di danneggiamento (D_c) che influenza direttamente lo strain rate a partire da uno strain rate iniziale derivante dal creep secondario (ε'_{co}):

$$\dot{\varepsilon}_c = \frac{\dot{\varepsilon}_{co}}{1 - D_c}$$

Il parametro di danneggiamento è legato da relazioni di tipo logaritmico allo stato di sforzo, alla temperatura e allo strain rate: lo strain rate risulta quindi legato al tempo mediante una andamento esponenziale (come descritto anche in figure 1 nella fase di creep terziario).

La valutazione, accanto alle deformazioni differite, di un parametro di danneggiamento, rendono il modello peculiare: per questo motivo non può essere approssimato, nelle verifiche, con uno dei pur numerosi modelli classici di viscosità usualmente presente nei codici di calcolo.

Per venire incontro alle esigenze dei suoi Clienti del settore, che hanno la necessita di condurre verifiche in accordo alle indicazioni

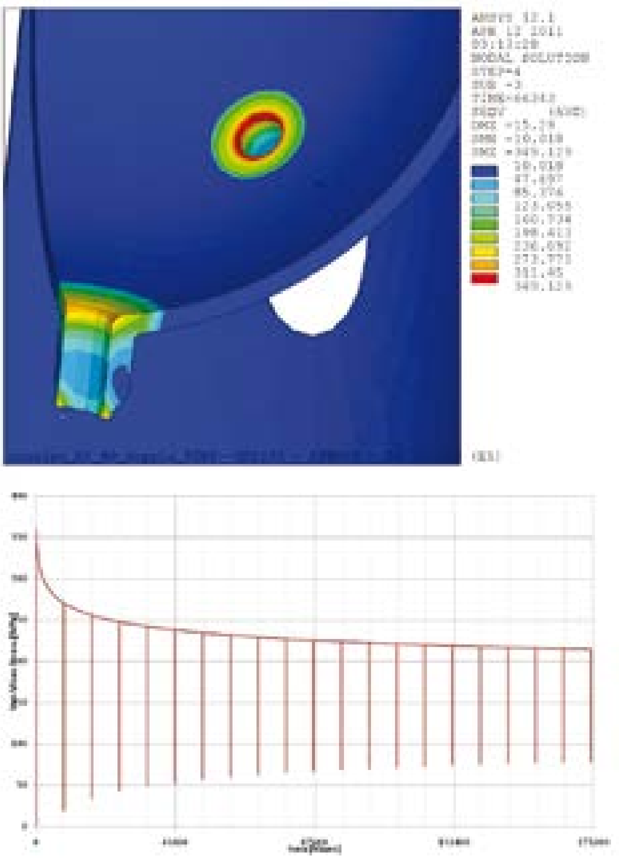


Fig. 2 - Esempio di calcolo secondo Code Case 2605 di due bocchelli in prossimità dell'attacco gonna. Il grafico evidenzia il rilassamento degli stress dovuto alla presenza del creep e all'aumento del danno associato, durante un vita di 20 anni. Si osservano gli effetti di scarico dovuti a operazioni di "Start Up" e "Shut Down" previste con cadenza annuale

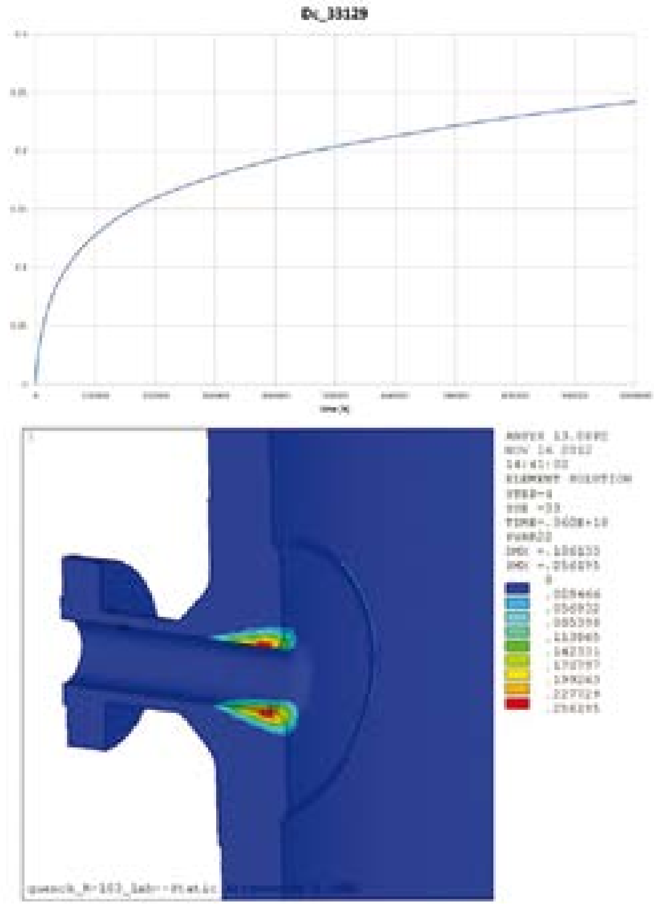


Fig. 3 - Danno accumulato nel punto più critico per un ciclo di 1000000 di ore (Dettaglio di un bocchello). Questa analisi permette di determinare la previsione di vita del componente per il solo effetto di creep e indicare al progettista su quale curva a fatica dovrà svolgere l'analisi accoppiata fatica/creep

ASME, Enginsoft ha quindi sviluppato un estensione alle funzionalità che permettere di aggiungere il modello MPC Omega alla libreria di modelli di creep presenti in ANSYS.

La procedura si basa su una user-defined routine, compilata in una libreria (*.dll) dedicata, che permette di inserire le valutazioni di plasticità e creep col modello MPC Omega richieste dal Code Case 2605 o dal FFS: le valutazioni risultano direttamente applicabili sui modelli ad elementi finiti già utilizzati per altre verifiche di stress analisys sui recipienti sotto esame.

La routine messa a punto su commesse di alcuni dei più importanti produttori italiani nel campo dei recipienti in pressione è estremamente flessibile e permette di introdurre direttamente dall' interfaccia ANSYS i valori caratteristici dei materiali oltre che i parametri numerici necessari per controllare la convergenza della soluzione e di gestire e post-processare tutte i principali risultati che caratterizzano il modello MPC Omega (dalle deformazioni a creep al danno accumulato).

Oltre all' implementazione della routine, Enginsoft è in grado anche di fornire formazione e assistenza all'utilizzo della stessa. Dal punto di vista operativo le analisi che utilizzano il modello MPC Omega prevedono sia semplici analisi di screening per la valutazione del FFS di recipienti in pressione ad analisi di estrema complessità e onerosità computazionale per la simulazione della vita completa (fino a 30 anni) dei principali dettagli costruttivi.

A titolo di esempio, le figure seguenti mostrano risultati di alcune analisi effettuate su un Hydrocracking Reactor realizzato in acciaio grado F22V, operante nei regimi di temperatura ricadenti nel range

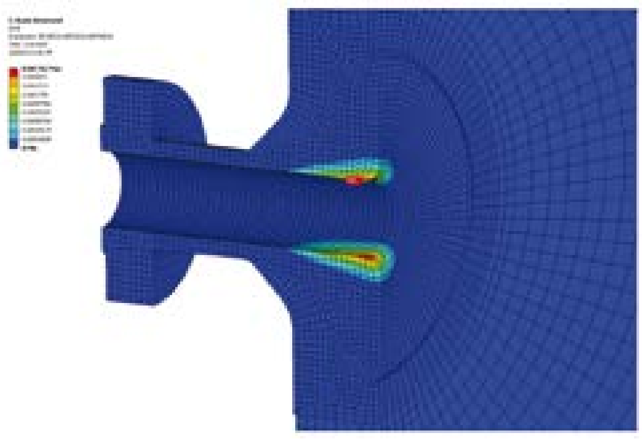
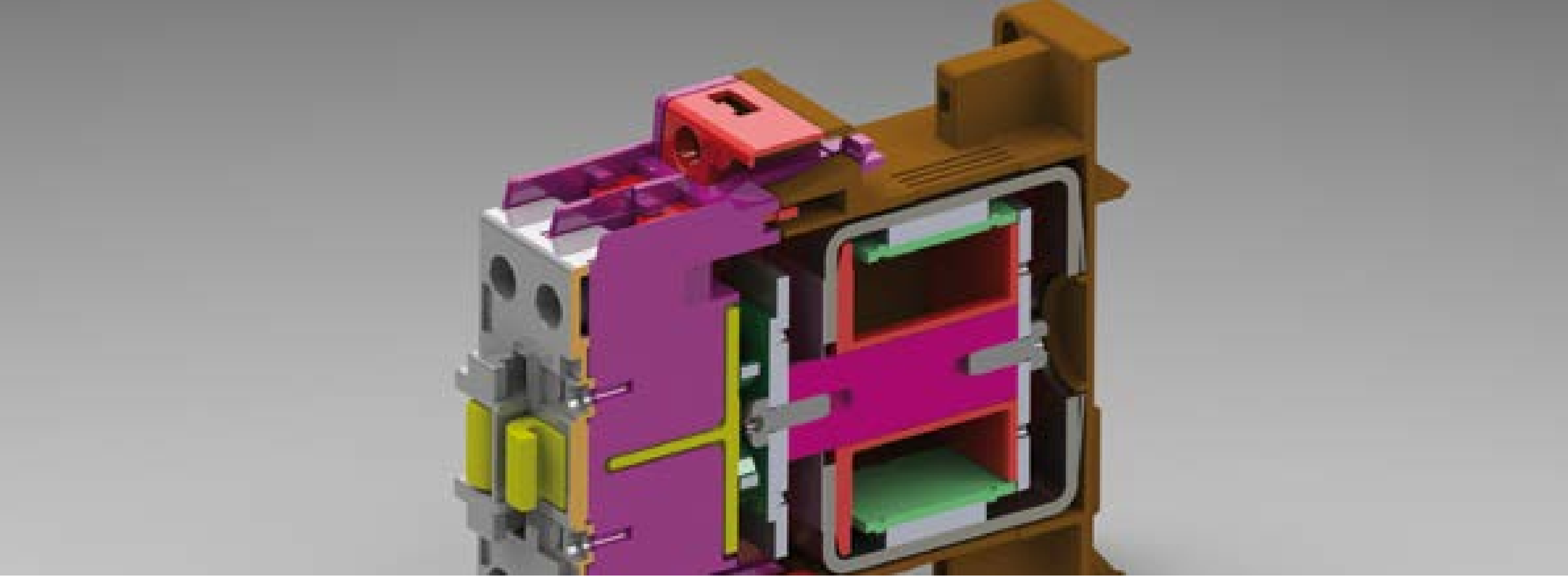


Fig. 4 - Esempio di deformazione totale accumulata alla fine dei cicli di vita di un bocchello. Da questo risultato, è possibile determinare, attraverso le curve a fatica ricalcolate nel cc2605, la vita a fatica del componente in presenza di fenomeni di creep

coperto dal Code Case 2605. Le analisi si riferiscono a modelli 3D di dettagli sensibili a concentrazioni di stress del reattore quali bocchelli e l'attacco gonna. In questa particolare applicazione è stata simulata l'intera vita di progetto (20 anni) del componente, incluse 20 fermate intermedia.

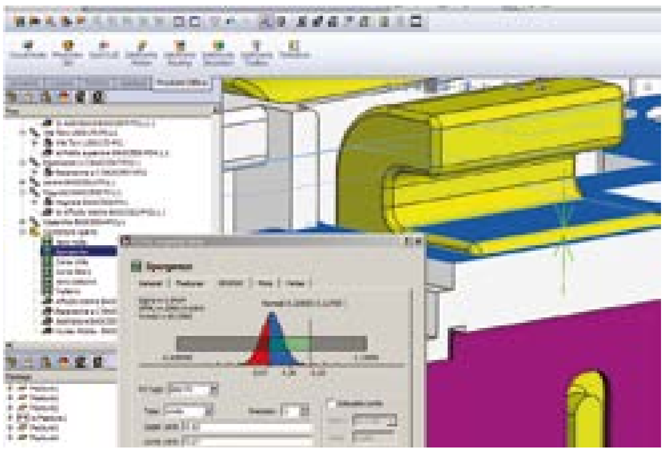
Per ulteriori informazioni:
Michele Camposaragna - EnginSoft
info@enginsoft.it



CETOL 6: un software innovativo per la rapida soluzione dei problemi derivanti dalla propagazione tridimensionale delle tolleranze

CETOL 6 è uno strumento software destinato a progettisti e responsabili della produzione e qualità, specificatamente concepito per individuare ed eliminare i problemi derivanti dalla propagazione degli errori intrinseci in tutti i processi produttivi.
Laddove un prodotto è costituito da più parti assemblate insieme, le variazioni dimensionali e di forma dei sotto componenti si combinano, talvolta in modo estremamente complesso, andando a modificare le proprietà complessive del prodotto da cui dipende la sua conformità.

Normalmente, il progettista definisce al CAD la geometria nominale dei componenti, ovvero la geometria che, idealmente, soddisfa ogni requisito di funzionamento. Successivamente, assegna opportune tolleranze a tutte le dimensioni. In ambito progettuale, una tolleranza è l'intervallo di variabilità entro il quale una proprietà geometrica si ritiene accettabile. Il loro utilizzo è necessario perché, in funzione del processo manifatturiero che verrà adottato, le dimensioni dei componenti reali presenteranno una variabilità più o meno accentuata attorno al valore nominale atteso. Questi fenomeni sono normalmente interpretati ricorrendo a strumenti statistici che sintetizzano indicatori molto comuni come la media, il tipo di distribuzione e la deviazione standard.



Se il prodotto finale è costituito da un singolo componente, lo studio della propagazione degli errori perde di significato e la sua accettabilità è verificata dal solo rispetto delle tolleranze. Viceversa, se il prodotto finale è un sistema assemblato multi-componente, occorre focalizzare l'attenzione sulle dimensioni complessive dalle quali dipende la capacità di assolvere la funzione per cui è stato progettato. Per comodità espositiva, indicheremo tali dimensioni con il termine di "quote funzionali". Affinché un prodotto assemblato sia conforme, è necessario che tutte le sue quote funzionali siano comprese tra i limiti prestabiliti dalle specifiche. Pertanto, una buona progettazione non potrà prescindere dall'analisi dei fenomeni che governano la propagazione degli errori dai componenti alle quote funzionali.
Quando il prodotto finale è formato da più componenti essi vengono tenuti insieme tramite incastri, viti, collegamenti filettati, incollaggi,

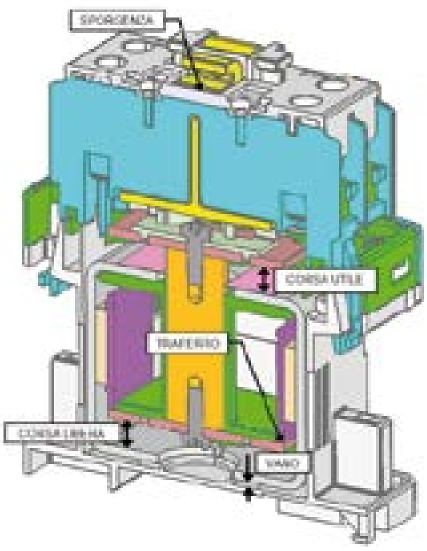
saldature etc. Indipendentemente dal tipo di collegamento, il posizionamento relativo (ovvero la posizione occupata da ciascun pezzo rispetto a quelli adiacenti) è definito da contatti che si instaurano tra superfici reali accoppiate. La configurazione reale del sistema assemblato non è mai coincidente con la configurazione nominale prevista. Infatti, la complessa combinazione delle proprietà geometriche (disperse) dei componenti, genera un'inevitabile dispersione sulle quote funzionali dell'assemblato. E' importante sottolineare che il risultato della propagazione degli errori è di difficile determinazione perché la tridimensionalità del problema presuppone fenomeni di amplificazione non trattabili con un classico approccio basato sulla combinazione lineare degli effetti.

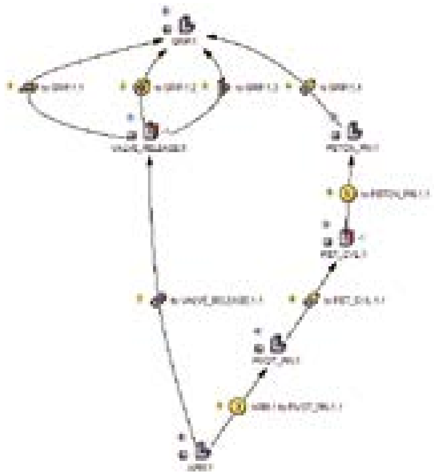
A causa di questa complessità nella formulazione rigorosa del problema succede molto spesso che, assemblando componenti conformi (cioè con dimensioni che singolarmente rispettano i limiti di tolleranza), si ottengano configurazioni finali con quote funzionali che non soddisfano le specifiche. Questo ribadisce che, quando si progettano sistemi multi-componente, l'assegnazione di quote nominali e tolleranze non può essere fatta senza uno studio mirato né, tantomeno, affrontando il problema in modo semplicistico.

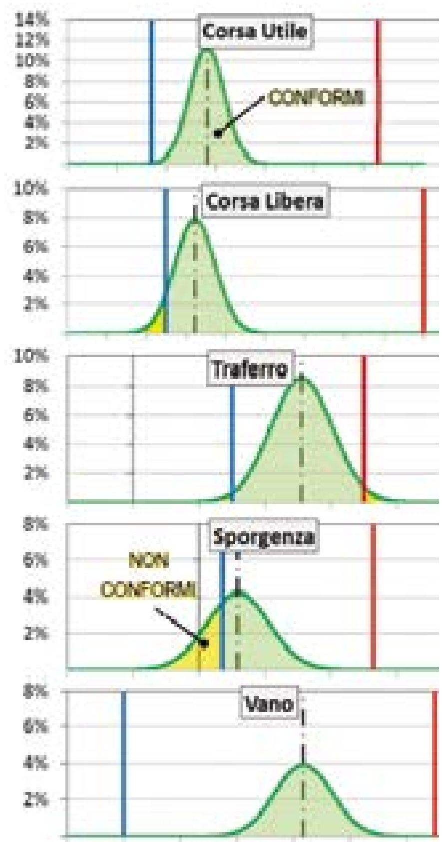
CETOL 6 nasce con l'intento di coprire in modo molto dettagliato questo genere di problematiche.

A differenza di altri strumenti meno evoluti, CETOL 6 affronta la propagazione degli errori tenendo in considerazione la tridimensionalità delle superfici reali, la possibilità che queste interagiscano in punti diversi in relazione agli errori, e dispone di avanzati strumenti statistici per analizzare il problema nel suo complesso. Inoltre, per facilitare la creazione dei modelli di analisi, possiede un'interfaccia completamente integrata con i sistemi CAD più diffusi (proEngineer/CREO, SolidWorks, CATIA V5, ecc.).

I risultati ottenuti da CETOL 6 consentono di individuare in poco tempo l'eventuale esistenza di problemi sulle quote funzionali di un sistema assemblato. Quindi, è possibile risalire immediatamente a quelle tolleranze che danno un effetto più marcato di altre sulle quote funzionali, consentendo in questo modo di intervenire laddove si otterranno i migliori risultati con il minimo sforzo. CETOL 6 è particolarmente versatile, perché può







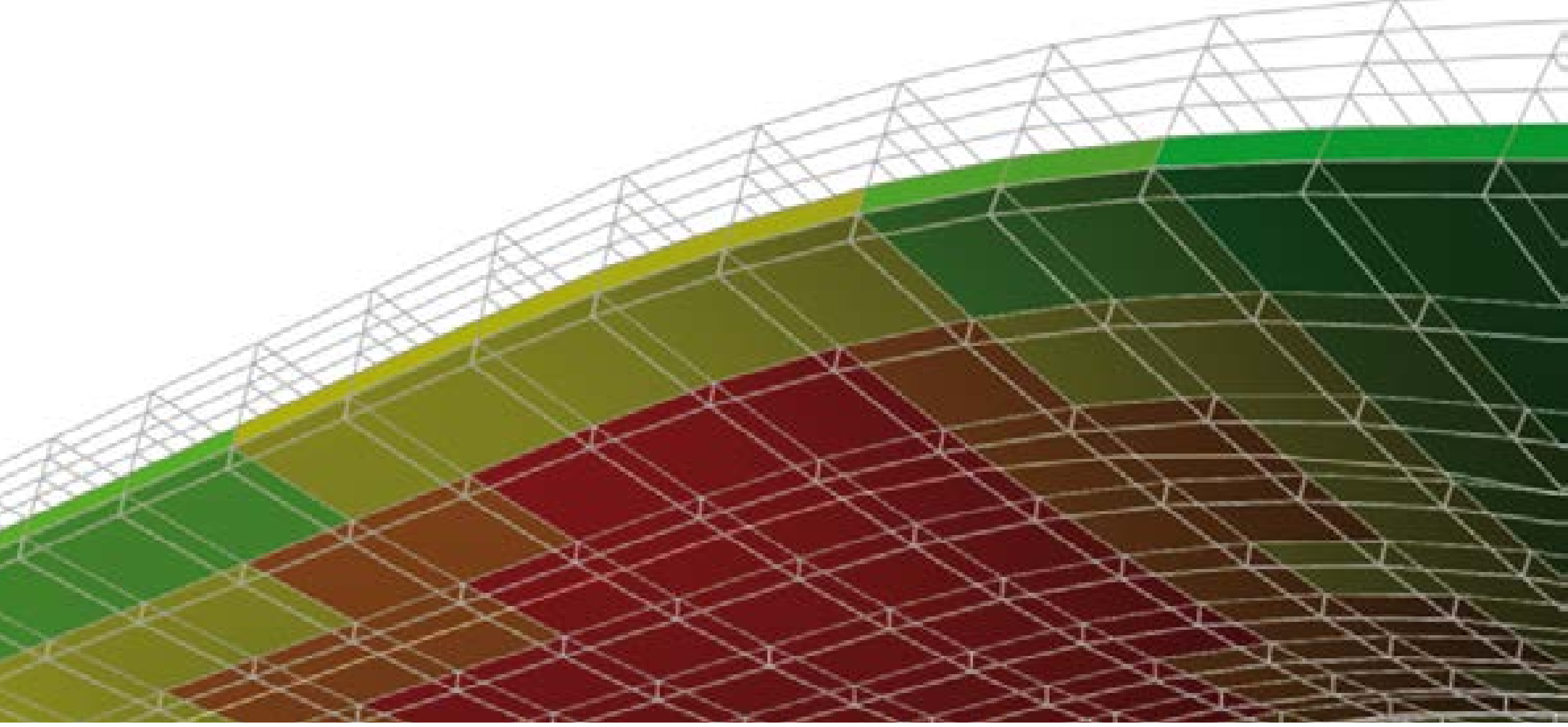
essere applicato sia in fase di progettazione con finalità di ottimizzare l'assegnazione delle tolleranze, oppure a valle della produzione per scegliere come risolvere eventuali problemi emersi.

L'ausilio offerto è di assoluto spessore. Infatti le tolleranze sono direttamente responsabili delle funzionalità del prodotto finito e sono, allo stesso tempo, connesse in modo diretto con l'applicabilità di processi produttivi più o meno raffinati (e quindi con i costi di produzione). Ne consegue che, in generale, la scelta delle tolleranze rappresenta un compromesso tra l'esigenza di contenere i costi di produzione e l'esigenza di garantire la funzionalità del prodotto finito.

L'importanza di uno strumento come CETOL 6 è ancora più evidente se si pensa che, in molti casi, è possibile risolvere una non conformità semplicemente agendo sul valore nominale di una quota, lasciando invariato il valore della tolleranza ad essa assegnata. Questo intervento, quando possibile, risolve il problema, mantenendo inalterata tecnologia produttiva e relativi costi.

Svariati sono i settori industriali dove i benefici di CETOL 6 possono essere ragguardevoli. Infatti, i problemi derivanti da una errata assegnazione delle tolleranze, possono avere forti ripercussioni economiche sia su piccole che su grandi produzioni di serie. Citiamo, a titolo di esempio, il settore automotive, quello aeronautico, l'elettronica di consumo, la produzione di dispositivi ottici, elettromeccanici, la robotica industriale, il settore biomedicale, e via dicendo.

Per maggiori informazioni:
Gianmaria Cusimano - EnginSoft
info@enginsoft.it



Le novità di ANSYS Composite Prep/Post versione 14.5

Il modulo ANSYS Composite Prep/Post, sin dalla prima versione rilasciata, ha profondamente modificato il modo di affrontare le problematiche di progettazione assistita per le strutture in materiale composito. ACP costituisce uno strumento che, mediante avanzate feature per il pre ed il post-processing di modelli complessi, è in grado di operare nell'ottica del "Concurrent Design", in base alla capacità di guidare il progettista nello sviluppo integrato del prodotto in materiale composito e del relativo processo produttivo.

La nuova versione 14.5 di ANSYS, rilasciata a novembre 2012, ha introdotto una serie di strumenti e funzionalità che consentono al progettista di lavorare in maniera sempre più efficiente, rapida ed accurata. Lo sviluppo dell'interfaccia di progetto Workbench e degli ambienti di modellazione e simulazione al suo interno (e.g. DesignModeler, Mechanical, …) permettono una gestione delle geometrie fino a 10 volte più rapida, un controllo efficace delle opzioni di meshing, delle interfacce di contatto e degli strumenti di post processamento dei risultati. Alla nuova versione del software ANSYS è seguito l'aggiornamento del modulo ANSYS Composite Prep/Post, che nella release 14.5 viene ulteriormente integrato con l'ambiente Workbench e potenziato con nuove feature di calcolo.

Mediante gli strumenti di calcolo introdotti nella nuova versione del modulo ACP, in fase di pre e post processing del modello in composito è possibile eseguire in maniera molto semplice una serie di operazioni che, viceversa, racchiudono un notevole significato dal un punto di vista ingegneristico. Seguendo una procedura di progettazione integrata processo/prodotto, è possibile simulare gli effetti dei processi tecnologici dell'Hand Layup e del Filament Winding su geometrie complesse anche in presenza di superfici con una considerevole doppia curvatura; in tal caso è possibile prevedere e con-

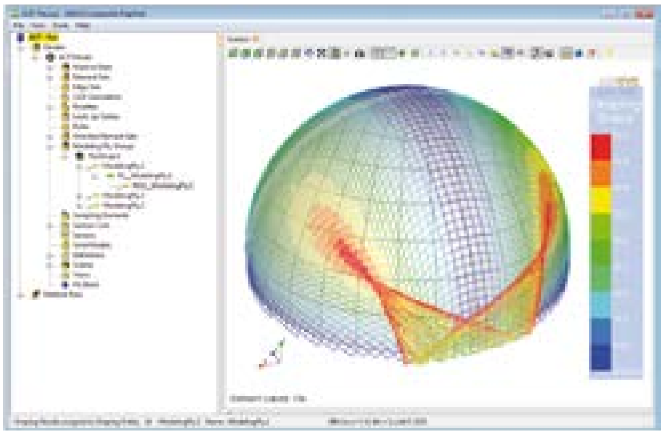


Fig. 1 – Simulazione di drappeggio e overlapping della lamine rinforzate sulla superficie a doppia curvatura

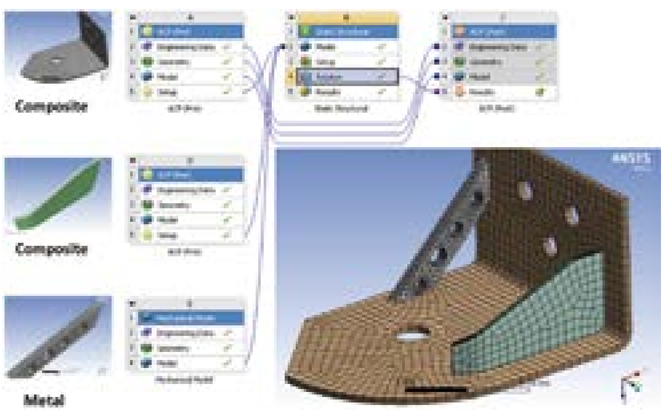


Fig. 2 – ACP integrato in ANSYS Workbench – Procedura per l'analisi di strutture in composito mediante modellazione 3D

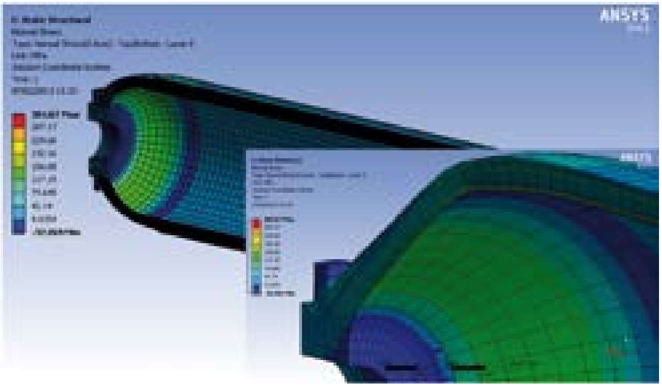


Fig. 3 – Analisi tensionale sul modello di un serbatoio in pressione realizzato per filament winding

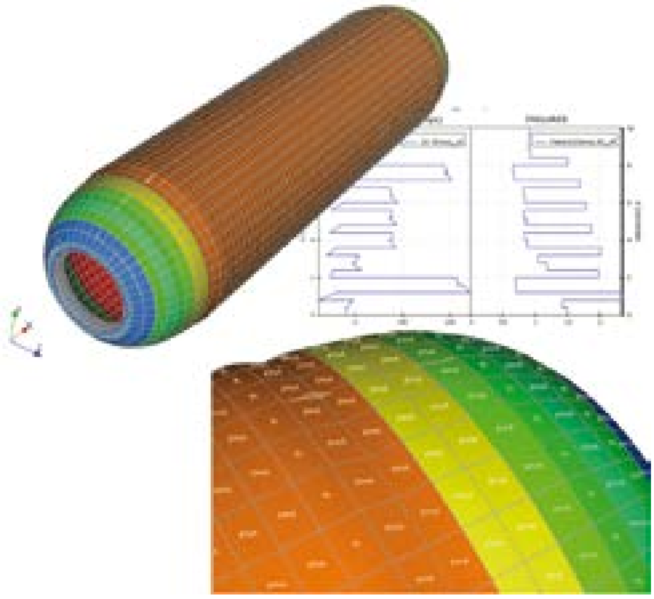


Fig. 4 – Analisi tensionale sul modello di un serbatoio in pressione – Distribuzione degli stress e dell'IRF nello spessore

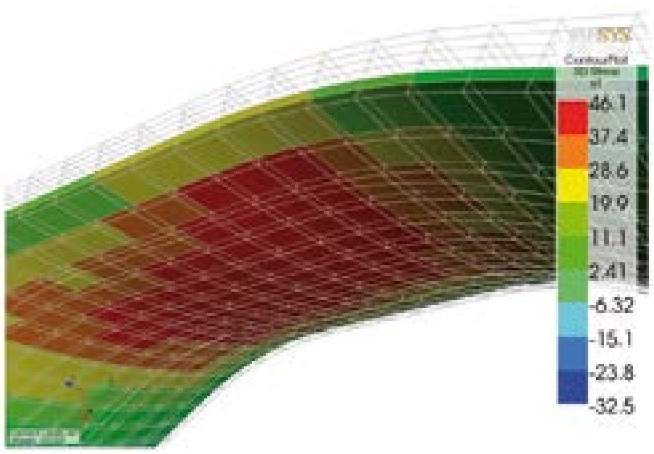
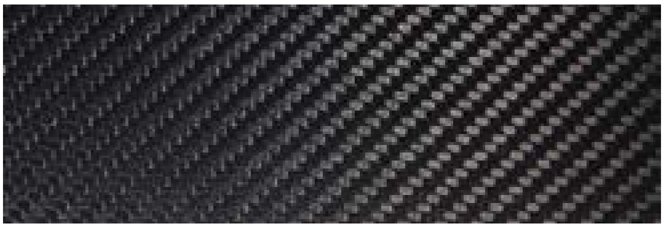


Fig. 5 – Analisi tensionale sul modello 3D – Distribuzione degli stress nella direzione di stesura delle fibre

trollare l'eventuale la formazione di grinze, inspessimenti e locali sovrapposizioni (overlapping), esaminando in fase di verifica numerica la variazione delle prestazioni strutturali in funzione della presenza di tali difetti produttivi.
Un considerevole passo avanti è stato compiuto nell'ambito della modellazione di strutture in composito con elementi solidi: i componenti 3D realizzati in materiale composito e quelli in materiale isotropo possono essere trattati separatamente, per essere successivamente combinati all'interno di un unico sistema assemblato. L'analisi dei risultati relativi all'intero modello ed in particolare ai componenti tridimensionali in materiale composito, viene realizzata mediante criteri di resistenza 3D appositamente elaborati per la previsione delle failure conditions sui compositi (delamination, core failure, debonding, …). Gli effetti del danneggiamento subito dal laminato possono essere analizzati mediante avanzate tecniche numeriche in grado di riprodurre la delaminazione degli strati rinforzati e la successiva propagazione interlaminare del difetto (Virtual Crack Closure Technique).

Infine la nuova versione del software consente di applicare le tecniche di sottomodellazione utilizzate in Workbench anche ai modelli ACP, di ridurre lo spazio occupato dai file dei risultati fino al 60% e di parametrizzare i modelli in maniera semplice operando direttamente da interfaccia grafica senza dover intervenire a livello di scripting. I parametri di input e di output definiti all'interno dell'ambiente ACP-Pre e ACP-Post sono automaticamente riportati all'interno della sezione "Parameter Set" di ANSYS Workbench, al fine di facilitarne l'elaborazione e la manipolazione per generare nuovi Design Point o per una successiva ottimizzazione.

Per maggiori informazioni:
Fabio Rossetti - EnginSoft
info@enginsoft.it



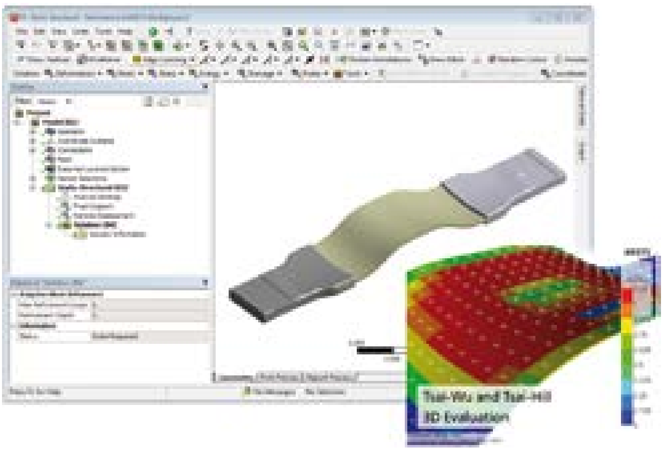
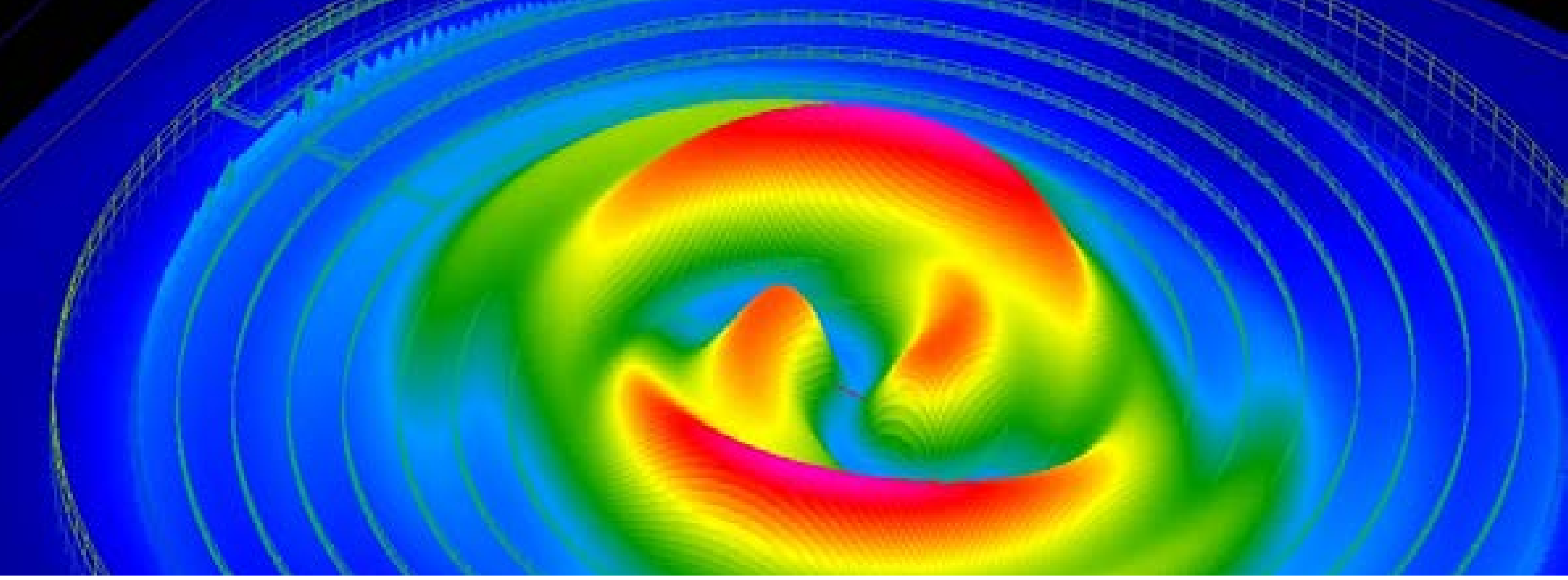


Fig. 6 – Post-processing dei risultati secondo la formulazione 3D dei criteri di Tsai-Wu e Tsai-Hill



Openeering has new success stories to tell:

Scilab in the crosshairs of industry

In these last months, Openeering registered a renewed interest from industries on tutorials and services. Industries are approaching Scilab with an increasing attention, captured by all the benefits of its use. In fact, the ongoing global crisis started in 2008 has forced companies to focus on efficiency and costs reduction pushing them to explore open source software as a possible alternative to closed source.
With the option given by Openeering and the access to a wide range of expertise to address technical challenges given by EnginSoft engineers, our customers can now refer to a single engineering services provider for CAE and applied mathematics. This is what we have done to support our customers in these last months.

In this article we want to tell the reader about one of our new success stories starring Scilab and the Openeering team.
The story is about how Scilab reduces, or even eliminates MATLAB® license costs, and this is particularly important when using parallel computation. This success story involves Thomatronik, a German company based in Rosenheim.

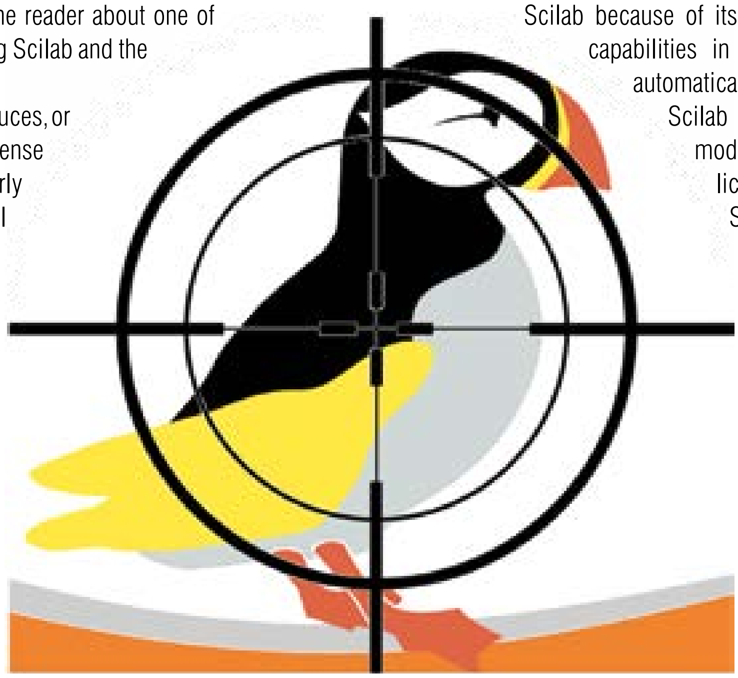


Fig. 1 - The puffin is Scilab logo and mascot. The puffin, as a symbol of the open source software, is now in the crosshairs of industries that are forced to focus on efficiency and costs reduction

Thomatronik

Thomatronik is a renowned supplier of superior products and services for industry and science over the last 25 years. Thomatronik offers three product segments with hardware, instrumentation / sensors and Finite Element Electromagnetic Computing software support, training and consulting.

Thomatronik is running multiple instances of their proprietary simulation software in parallel, where, at the end of each simulation run, a Fourier analysis has to be performed using a MATLAB® script. In mathematics, Fourier analysis is the study of how a general function may be approximated by the sum of simple trigonometric functions, in particular by sine and cosine. The decomposition of a function in a sum of sines and cosines is better known as "Fourier Transform".
Thomatronik recognized that Scilab was their option to use parallel simulation without increasing the cost of commercial license, as each individual simulation would require a separate license. They chose Scilab because of its fast and efficient computational capabilities in addition to the possibilities to automatically call parallel instances of the Scilab routine in background batch mode with no limits of concurring licenses. Thomatronik realized that Scilab differs from other products in the market (e.g. Octave, Maxima) because of its maturity. Actually, Scilab development and maintenance are carefully managed and controlled by the Scilab Enterprises.

Thomatronik has an extensive experience in signal processing and Fourier analysis, but they were usually doing that with MATLAB®. To help solve the problem, the Openeering team has implemented a simple Scilab module that reads an

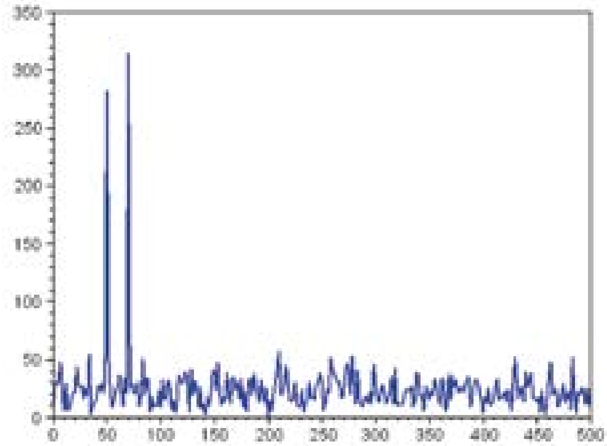
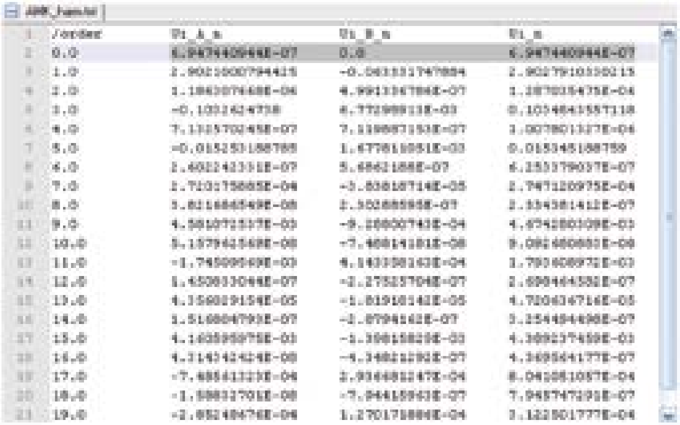


Fig. 2 - Frequency components of a signal containing pure frequencies at 50 and 70 Hz. FFT example in Scilab. To reproduce the plot of this example see box at the end.



/order	U1_A_n	U1_B_n	U1_n
0.0	6.9474409944E-07	0.0	6.9474409944E-07
1.0	2.9021000794425	-0.063331747884	2.9027910330215
2.0	1.1863076468E-06	4.9913367868E-07	1.2870354875E-06
3.0	-0.1032424738	6.772989113E-03	0.1034641557118
4.0	7.1325702458E-07	7.1198871538E-07	1.007801327E-06
5.0	-0.0152531088785	1.6778110051E-03	0.015345188759
6.0	2.4022423331E-07	5.6862188E-07	6.253379037E-07
7.0	2.7231758885E-04	-3.83818714E-05	2.747120975E-04
8.0	3.8216868549E-08	2.302885958E-07	2.334381412E-07
9.0	4.5810721537E-03	-9.28800743E-04	4.6742800309E-03
10.0	5.157942569E-08	-7.408141818E-08	9.082480883E-08
11.0	-1.745006669E-03	4.143308163E-04	1.793608972E-03
12.0	1.450633044E-07	-2.27523704E-07	2.690464532E-07
13.0	4.156029154E-05	-1.81918142E-05	4.720634716E-05
14.0	1.510804793E-07	-2.0794142E-07	3.254494498E-07
15.0	4.160595975E-03	-1.39815829E-03	4.389237459E-03
16.0	4.314342424E-08	-4.34821292E-07	4.369564177E-07
17.0	-7.68541333E-04	2.934681247E-04	8.041081057E-04
18.0	-1.58832701E-08	-7.94415963E-07	7.945747291E-07
19.0	-2.85248476E-04	1.270175884E-04	3.122501777E-04

Fig. 3 - An example of the output file with the list of coefficients for an increasing order of the polynomial.

ASCII file containing the signal as time and voltage data and carries out the Fourier analysis of the data and writes the coefficients (sine, cosine and amplitude) and the order in an output file as reported in fig. 3. The module can be executed in batch, by providing the name of the input ASCII file.

This enabled Thomatronik to carry out the Fourier analysis from their proprietary simulation software in a no-window mode and highly parallel fashion, without the need of expensive commercial software licenses.
EnginSoft provided Thomatronik with a solution based on a Scilab's script with its source code, and examples. The general benefits for companies adopting these solutions include:

- Better control: using open source lets companies be in control of their business
- Higher flexibility: the final solution can be modified by companies to suit the specific needs of their business in the future. Moreover there are no restrictions to number of seats or concurring licenses, this enables Thomatronik to install the solution in an unlimited number of computers.
- Higher Return on Investment: open source software can be obtained and used with little to no up-front costs. Companies only pay for the support they actually need, and most importantly only when they need it.

Openeering mission is to spread the use of Scilab and Xcos within industries. With the support of the wide range of expertise offered by EnginSoft, the Openeering team can help any company in creating tailored toolboxes to address several technical challenges including, but not limiting to, mathematical modeling, numerical simulation, optimization, statistics and data mining.

Useful links

http://www.thomatronik.de
Thomatronik website

http://www.openeering.com/made_with_scilab
Success stories on the Openeering website

http://www.scilab.org
To download your Scilab installation package

For more information
Silvia Poles, EnginSoft
info@enginsoft.it

Conclusions

An increasing number of industries are looking at the world of open source software, and Scilab is now a robust, flexible and low-cost alternative to MATLAB®. The success story presented in this paper is the evidence of this statement. Thomatronik obtained a solution to his peculiar problem in a short time and on a limited budget. Moreover Scilab have no restrictions to number of seats or concurring licenses and this enables to install the solutions in an unlimited number of computers. The companies only pay for the support they actually need, and are now free to use the toolbox and contact the Openeering team only when another extra consultancy is needed.

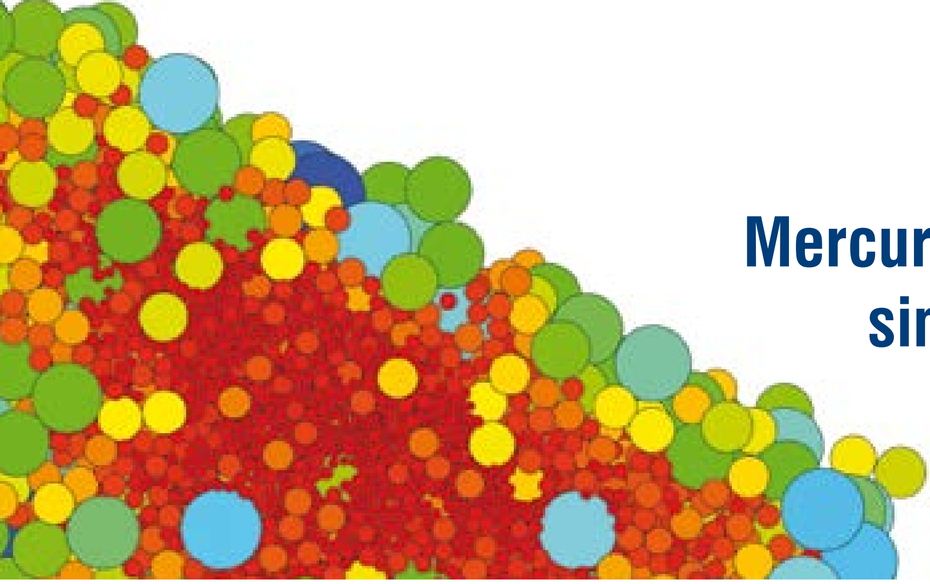
Try it yourself!

```
//Example from the Scilab manual
//Further examples are available in the Scilab Help
//-----------------------------------
//Frequency components of a signal
//-------------------------------------
// build a noised signal sampled at 1000hz  containing  pure frequencies
// at 50 and 70 Hz
sample_rate=1000;
t = 0:1/sample_rate:0.6;
N=size(t,'*'); //number of samples
s=sin(2*%pi*50*t)+sin(2*%pi*70*t+%pi/4)+grand(1,N,'nor',0,1);

y=fft(s);

//s is real so the fft response is conjugate symmetric
//and we retain only the first N/2 points
f=sample_rate*(0:(N/2))/N; //associated frequency vector
n=size(f,'*')
clf()

plot(f,abs(y(1:n)))
```



Mercury-DPM: Fast particle simulations in complex geometries

Mercury-DPM is a code for performing discrete particle simulations. That is to say, it simulates the motion of particles, or atoms, by applying forces and torques that stem either from external body forces, (e.g. gravity, magnetic fields, etc…) or from particle interactions. For granular particles, these are typically contact forces (elastic, viscous, frictional, plastic, cohesive), while for molecular simulations, forces typically stem from interaction potentials (e.g. Lennard-Jones). Often the method used in these packages is referred to as the discrete element method (DEM), which was originally designed for geotechnical applications. However, as Mercury-DPM is designed for simulating particles with emphasis on contact models, optimized contact detection for highly different particle sizes, and in-code coarse-graining (in contrast to post-processing), we prefer the more general name discrete particle simulation. The code was originally developed for granular chute flows, and has since been extended to many other granular applications, including the geophysical modeling of cinder cone creation. Despite its granular heritage it is designed in a flexible way so it can be adapted to include other features such as long-range interactions and non-spherical particles, etc.

Why a new simulation code?

There are many open-source particle simulation packages, so the question arises of why another? Mercury-DPM was originally started as a joint collaboration between the Multi-Scale Mechanics (MSM) and the Mathematics of Computational Science (MaCS) groups (before 2011 they were called the Numerical Analysis and Computational Mechanics) at the University of Twente in 2009. The idea was to develop a code that could be used alongside the existing MaCS group continuum solver hpGEM (http://wwwhome.math.utwente.nl/~hpgemdev/) to approach problems using various multi-scale computational methods. Around the same time Vitaly Ogarko and Stefan Luding developed an advanced contact detection method: the hierarchical grid. This novel algorithm is quicker than existing methods for poly-dispersed flows (and still the same speed for mono-dispersed). So the idea of a new simulation code that had three core design aims was born:

1. It should be easy to use with minimal C++ knowledge.
2. It should be built around the new hierarchical grid detection method.
3. It should be able to generate accurate continuum fields that could be used with/alongside continuum solvers.

Actually, the name of the code emanates from the contact detection method: hierarchical grid → Hgrid → Hg → Mercury.

Features

Since it was first started it has evolved and gained many novel features. The main features include:

1. Of course, the hierarchical grid: The neighborhood search algorithm to effectively compute interaction forces, even for highly poly-dispersed particles.
2. Built-in coarse-graining statistical package: it has an in-built advanced statistics package to extract continuum fields such as density, velocity, structure and stress tensors, either during the computation or as a post-processing step.
3. Access to continuum fields in real time: The code can be run in live statistics mode, which means it can respond to its current macroscopic state. An illustrative example of using this would be a pressure-release wall, i.e., a wall whose motion is determined by the macroscopic pressure created by particle collisions and moves such that its pressure (not position) is controlled.
4. Contact laws for granular materials: many granular contact force models are implemented, including elastic (linear or Hertzian), plastic, cohesive, sintering (temperature/pressure/time-dependent), and frictional (sliding/rolling/torsion) forces.
5. Simple C++ implementation: Mercury-DPM consists of a series of C++ classes that are flexible, but easy to use. This allows the user to generate advanced applications with only a few lines of code.
6. Handlers: The code has handlers for particles, walls and boundaries. Thus, each object type has a common interface, even though individual objects can have completely different properties. This also makes it easier for the user to create new objects.
7. Complex walls: The code not only supports simple flat walls, but also axial-symmetric, polyhedral and helical screw walls are available. Additionally, due to the handler interface it is easy for more advanced users to define new types of walls themselves.
8. Specialized classes: Many specialized classes exist that reduce the amount of code required by the user to develop standard geometries and applications. Examples include chute flows, vertically vibrated walls and rotating drums.
9. Species: Particles and walls each have a unique species, which

is hidden for basic use of the code; however, this feature can be enabled by a single function call. Different particle properties for each species and different interaction forces for each pair of species can then be defined, allowing the simulation of mixtures.

10. Self-test suite and demos: Mercury-DPM comes with a large (over 100) self-tests and demo codes. These serve two purposes: 1) they allow us to constantly test both new and old features so we can keep bugs to a minimum; 2) they serve as tutorials, for new users, of how to do different tasks.
11. Simple restarting: every time a code is run (and at intervals during the computation) restart files are generated. Codes can be restarted without recompilation simply by calling the executable again with the restart file name as an argument. Also the restart files are complete in the sense that they contain all the information about the problem. In this way, small changes can be made (e.g. with the individual particle density or the coefficient of restitution) and the simulation can be rerun without the need for recompilation of the code.
12. Visualization: The particles output can be visualized easily using the free package VMD (visual molecular dynamics, http://www.ks.uiuc.edu/Research/vmd/).
13. Parallel: Currently a parallel-distributed version of the code is under development using MPI and this version should be publicly available shortly.

Simple C++ implementation and handlers

Mercury-DPM is a very versatile, object-oriented C++ code, which means new applications can be developed rapidly and easily. It has been tested for several Linux distributions and Mac OS. It consists of a core (kernel) that contains a series of C++ classes onto which users can quickly build to develop their own application (driver). The base class, Mercury3D, is flexible and contains the basic functionality to define a simulation. Using this class, the users specify the particulars of their simulations (initial positions, inflow, outflow, walls, interaction parameters, etc.) in a single driver file, which calls the kernel to perform the simulation. In addition to the flexible base class many higher-level, more powerful classes exist, which are tailored for common problems. A typical example would be the class Chute. This automatically defines a bottom that can be smooth or rough (of which we have three different types), an inflow boundary, outflow conditions, sidewalls, etc. and gives the user new access functions to perform standard tasks; for example: set_ChuteAngle (which automatics rotates the gravity vector), and set_InflowHeight (which changes the height of the particle layer at the entry to the chute). These common functions allow the simple setup of chute flow problems in just a few lines of code. Many other high-level classes exist and a full up-to-date list can be obtained from the website, http://www2.msm.ctw.utwente.nl/MercuryDPM. As the code is fully object-oriented, many of the classes build on each other adding extra levels of functionality. An example would be the class ChuteWithHopper, which replaces the inflow conditions in the original Chute class with a more complicated hopper construction. In addition, it adds new access functions, which allow the hopper properties to be set. Due to the object-oriented nature of the code it is easy for users to change a driver code from one class to a similar one. For example to change a Chute problem to a ChuteWithHopper problem all the user has to do is change the class he includes at the top of the code and replace the access functions like set_InflowHeight to the hopper equivalent i.e. set_HopperWidth, set_HopperHeight, etc. All the code defining the geometry and dealing with particle properties does not have to be changed and has exactly the same interface.

Another key feature of the Mercury-DPM design is the idea of handlers. There are three handlers in Mercury-DPM: Particle, Wall and Boundary. Handlers mean that all items of the same basic type are stored in one place. This has several nice advantages the primary being the flexibility, i.e., each particle, wall, etc., can have completely different properties but as long as the basic properties are defined the code can deal with the item. The user does not have to look after the walls, particles and boundaries themselves; they only have to create them. For example, to add a new particle to the simulation the user defines the properties of the new particle and passes them on to the ParticleHandler, then the code does the rest. The user does not need to know anything about other particles that have previously been created. The handler can also be queried via its access function to obtain information like the number of particles currently in the simulation, the smallest particle, etc.

Applications

Here we will illustrate some of the features of Mercury-DPM via applications that have already been developed in the package.

Poly-dispersed segregation in a rotating drum (S. Gonzalez, S. Luding, A.R. Thornton)

One of the most fascinating properties of granular matter is the ability of appropriately driven mixtures to separate into their individual components, despite the apparent lack of energetic or entropic advantages of a segregated state. This often produces brilliant patterns that give rise to a number of interesting problems in nature and difficult challenges for the powder compressing. The segregation of a binary mixture contained in a partially filled, horizontal, rotating drum is an extensively studied problem of this class; one with obvious industrial importance. One of its most beautiful characteristics occurs when it segregates in the radial direction, producing a core rich in small particles surrounded by an outer layer of mainly big particles, and depending on the angular velocity, rich patterns of segregation. Despite the great number of studies involving two-components systems, poly-disperse systems remain mainly unexplored, although they are more the rule than the exception in nature. The importance of these systems for industry is obvious; from a theoretical and computational point of view, they present various and difficult challenges.

One of the key reasons why poly-dispersed flow has not been investigated in the past is the computational cost. Traditionally, particle simulation codes use a linked list system for contact detection. This method has a complexity of order N for mono-dispersed flows, where N is the total number of particles. This means that if you double the number of particles the total computational time doubles. However, for poly-dispersed flows this nice scaling is lost and in the extreme limit of one very large particle and the rest containing small particles, the complexity

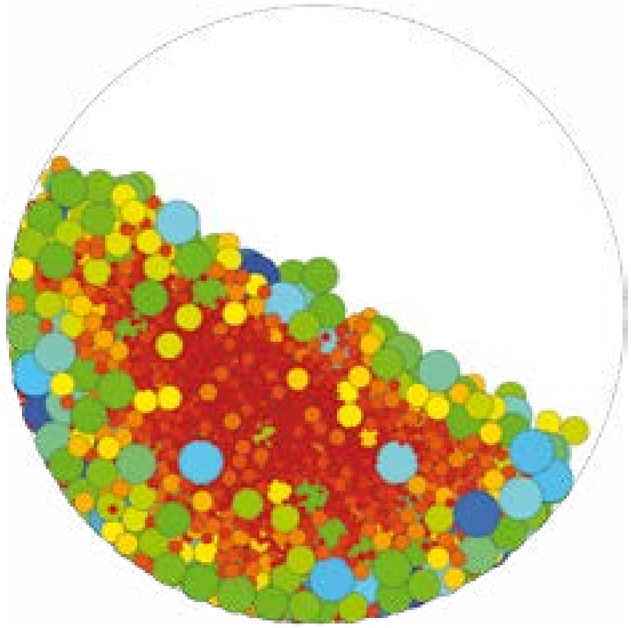


Fig. 1 - Poly-dispersed segregation in a rotating drum. Colour denotes particle size

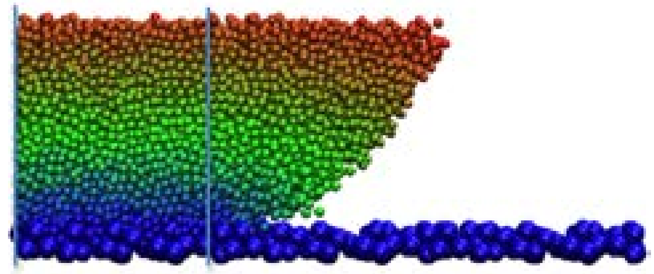


Fig. 2 - Illustration of the Mercury inflow maser. Colours indicate particle speed, with blue low and red high speed. Lines indicate the modified periodic boundaries.

becomes order N2. Yet due the hierarchical grid contact detection algorithm that forms Mercury-DPM's heart this problem is still order N within Mercury-DPM. This is why highly poly-dispersed and wide-size distributions can be easily tackled for the first time, in an open source environment.

Our simulations consist of spherical particles with different size distributions. Fixing particles to the surface of a given geometry makes the walls of the rotating tumblers. An easier way is to define finite walls; this is done in the driver. Turning the angle of gravity simulates the rotation of the tumbler. This makes the simulation with finite walls easier, and since the speed of rotation is quite low, the approximation is valid. For higher speeds, centrifugal forces have to be considered or the walls moved (which is possible within Mercury-DPM). Figure 1 shows an example of one of these poly-dispersed simulations. In this simulation every particle is of a different size, with a uniform volume distribution. The color represents the size of the particles with red the smallest and blue the largest. The ratio of the smallest to largest particle is ten to one in this simulation. The image is taken after two revolutions of the drum, and a strong segregation pattern can be observed with the small particles located in the center of the drum.

Granular flow through a contraction (D. Tunuguntla, A.R. Thornton, T. Weinhart, O. Bokhove)

As a stepping-stone towards analyzing complex granular flows in industry, we analyzed flow in an inclined channel with a contraction. In order to simulate steady-state flow through the contraction, the flow should be in steady state when entering the contraction. Regular inflow conditions such as the insertion of particles at the boundary or through a hopper would require us to simulate a large stretch of flow before the contraction to obtain steady flow at the beginning of the contraction. In order to reduce the computational costs to a minimum, a new special type of inflow has been designed that produces steady uniform flow directly at the contraction entrance. This is done using a small periodic box in the inflow regions, the downstream wall of which both mirrors and transmits the particles into the main chute. That is, each time a particle moves through the downstream periodic wall, a copy is created which ignores the periodic walls and thus flows into the contraction. This inflow type was named maser, as it acts as a material laser, creating a steady uniform inflow of particles. Meaning that a small cheap steady-state periodic-box simulation can be used to seed the much larger simulation through the contraction. Details of this kind of inflow will be presented in a later publication. An illustration of this inflow is shown in Figure 2, here the flow is visualized in VMD (visual molecular dynamics, http://www.ks.uiuc.edu/Research/vmd/); Mercury-DPM contains wrappers to view its output in this package.

Mercury-DPM contains an implementation of arbitrary convex polyhedral walls. These walls have been carefully designed to ensure that the collision with each face, edge, or corner of the wall is treated correctly. The main difficulty here is to determine the nearest face, edge or corner, and the normal direction of each collision. In a particle-face collision, the normal always equals the face normal; whereas, the normal of an edge-particle collision depends on the position of the particle with respect to the edge; finally, the collision with a corner is equivalent to colliding with an infinite mass particle.

Figure 3 shows a simulation of the granular flow through a contraction formed by two polyhedral walls. Once the particles flowing down the channel enter the contraction, jumps/shocks

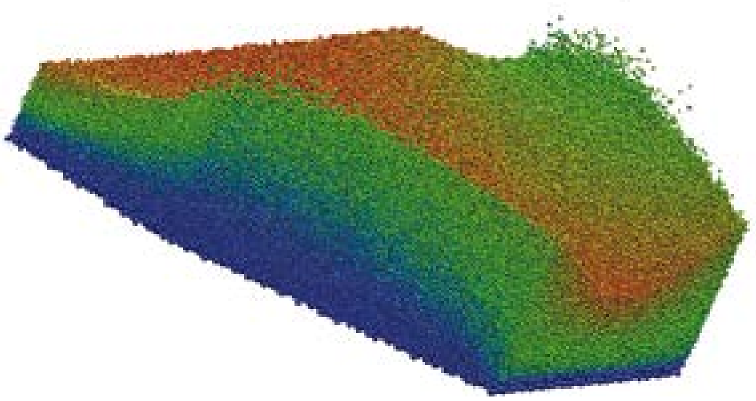


Fig. 3 - 380,420 particles flowing through a contraction. Colours indicate particle speed.

in depth and velocity profiles are observed. The interaction of the particles with the sidewalls of the contracting channel can be seen in Figure 3, where the colour denotes the speed. The blue region illustrates the jump in the velocity profile of the flow.

Granular jet impacting of an inclined plane (R.H.A. Fransen, A.R. Thornton, S. Luding, T. Weinhart)

Here, we simulate a granular jet impacting an inclined plane using Mercury-DPM. This problem was first investigated both experimentally and via the continuum approach by Johnson and Gray.

The novelties in the implementation are the construction of inflow conditions through a funnel and the modeling of a rough surface. Finally, the depth-averaged height and velocity are extracted from the simulation using our coarse-graining toolbox.

To obtain a jet of particles, a funnel is created using fixed particles placed onto a conical shape. Particles are inserted into the top third of the funnel whenever a free space is detected, see Figure 4 top left. The roughness of the funnel wall is necessary to create a velocity profile that keeps the developing jet from spreading.

To obtain the strong frictional effects observed in experiments, the plane needs to be rough as well. Therefore, a disordered layer of fixed particles is created. To prevent particles from falling through, a planar wall is placed below the fixed particles. Using the frictional rough walls allows us to observe similar profiles of the impact and fast-flowing zones as in the experiments, see bottom left and right in Figure 4.

Particles are removed from the simulation when they reach the end of the plate. This leads to a low pressure at the outflow, which can affect the flow on the plate. In Mercury-DPM the user can define a removal condition. If this is set to be a few cm below the plate, that is, when the particles are in a free flowing jet, off the end of the plate, it has been shown not to have an affect on the main flow.

One of the major novel features of Mercury-DPM is its coarse-graining toolbox, which constructs a continuous macroscopic field from the discrete particle data. This toolbox can be both run as either a post-processing step or live during the simulation. Careful attention has been paid to the boundary areas, and this package is even able to produce continuum fields within one particle diameter of a boundary. Examples of the results of the course-graining package for the jet problem are shown in Figure 5.

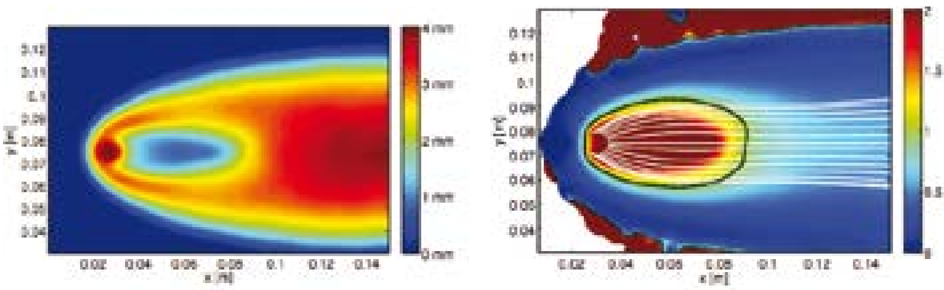


Fig. 5 - Course grained macroscopic fields created using Mercury-DPM's coarse-graining toolbox. Left shows the height of the flow in millimetres and right the local Froude number of the flow. The white lines indicate velocity streamlines; the black line indicates the location of a hydraulic jump/shock.

In order to obtain the height of the flow we assume that the density of the flow is constant over height, and that the flow is steady and uniform enough to have a lithostatic stress profile, see Figure 5. Thus, the height can be defined using the depth-averaged stress and density as plotted in Figure 5. Once the height is known, a depth-averaged velocity and the Froude number can be defined. A Froude number larger than unity denotes supercritical flow, otherwise the flow is subcritical. This allows us to determine the location of the shock (black line in right panel of Figure 3).

Screw feeder and conveyor (D. Krijgsman)

The final feature of Mercury-DPM we will illustrate in detail is the helical screw. This highlights the flexibility of the versatile handlers, they are common in many industrial apparatuses. The difficulty of these simulations lies in the interaction between the screw and the particles. The approach that is used in most similar particle simulation packages is to triangulate the screw and do collision detection between the particles and small segments of the screw. The major disadvantage of this method is that for accuracy the single screw element has to be divided into a large number of triangles. All possible combinations of these triangles with the particles have to be checked for contacts, resulting in high computational costs. To circumvent this, in Mercury-DPM the screw is modeled as a single parametric surface.

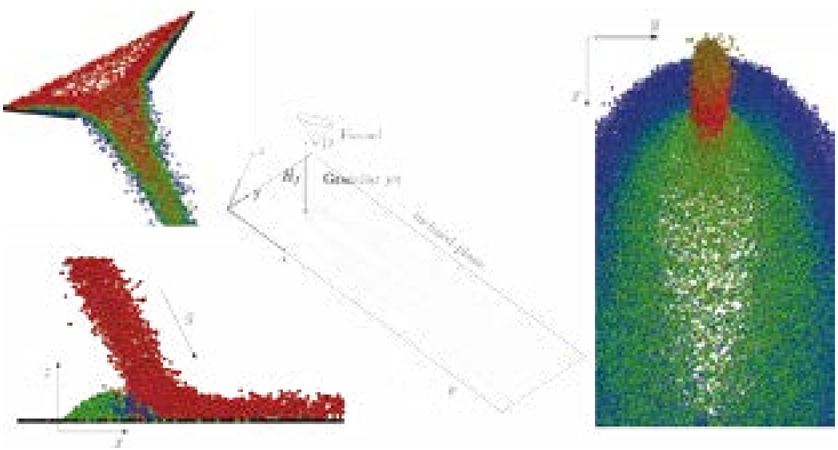


Fig. 4 - Top left shows the flow in the hopper, bottom left the impact region, middle schematic of the original experiment, right the top view of the full particle simulation (~500k particles). Black particles indicate fixed particles; all other colours indicate speed, with blue low and red high speed.

In Mercury-DPM the screw is treated as 'just another wall' so all the user has to do is to create a screw and pass this screw to the WallHandler. The code automatically deals with the collision detection and

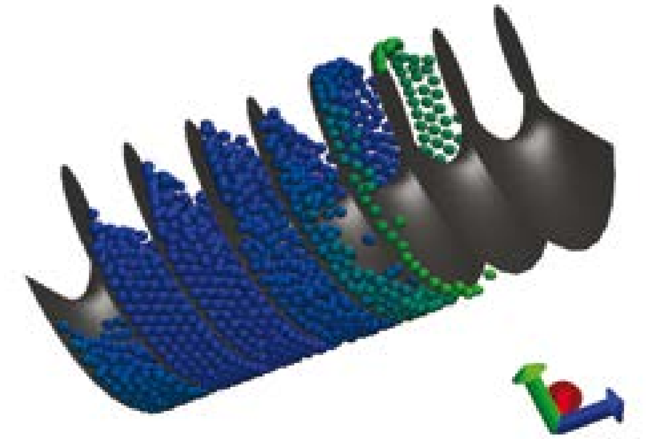


Fig. 6 - Snapshot of the screw conveyor simulation coloured by particle velocity. This transports the particles trough the tube. Colours indicate particle speed.

even rotation of the screw, if the user calls the move method. The screw is defined by a length, a maximum radius, the total number of twists, and a blade thickness. The mathematics of the definition of the surface will be omitted here, but they can be found in the full documentation of the code.

To check for collisions between a particle and the screw we have to find the point on the screw with minimum distance to the particle. This minimum is not analytically defined and Newton's method is used to quickly iterate to it. We then use this minimum to check if the screw and particle are in contact. Again, full details of this process can be found in the documentation of the code. The important points are: this process is invisible to the user; it is quicker than the triangulation method (if more than a hand full of triangles are used); it is more accurate, even in the limit of a large number of triangles; and, this method has no artificial numerical constants i.e. the number of triangles used for approximation of the actual screw.

Two standard industrial applications are used to illustrate the screw: Figure 6 shows an example of a screw conveyor, in this case, as the screw turns and particles are transported along its length, i.e., from left to right. In industry, screw conveyors are often used to transport particles to the next processing step. Figure 7 shows a screw feeder simulation, where the screw is positioned in a box, with a circular tube attached at the front end. The purpose of the feeder is to push the particles from the container into the tube to possibly feed another machine. Industrial apparatus simulations are able to provide detailed information on the flow inside the machine, which are difficult to obtain from experiments. With this detailed information one is able to investigate the optimization of these processes.

Parallel Mercury-DPM
(A. te Voortwis)

As the number of particles in a system increases it becomes unavoidable to solve the problem in a parallel manner. Therefore a parallel version of Mercury-DPM is currently under development. The implementation consists of a spatial domain decomposition in which the simulation domain is split up into several smaller domains, each of which is simulated in a separate process, such that each process can be seen as a 'standalone' simulation. This approach allows for the parallel implementation to be very transparent; it is simply a layer between the driver-codes and the kernel. The necessary communication (i.e. particles moving from one domain to another) between the different domains is done through the MPI protocol, and the communication overhead is minimized by ensuring that, in general, domains only communicate with their direct neighbors.

This approach ensures the proper scaling of the performance with an increasing number of processors. The bottleneck in this implementation would be the output of the result data, since this traditionally requires the sending of all data to a single process which writes the output. To overcome this issue, the HDF5 binary file-format is used for the file output because the library of this file-format allows each process to write its data in parallel, minimizing the overhead. The combination of the flexible spatial domain decomposition and the parallel file-output ensures that Mercury-DPM scales very well from an average desktop PC up to large scale parallel high performance computer systems. With the introduction of the new HDF5 format, also the serial code has to undergo some significant changes. For this reason, the launch of the parallel version is due for the next major release of Mercury- DPM.

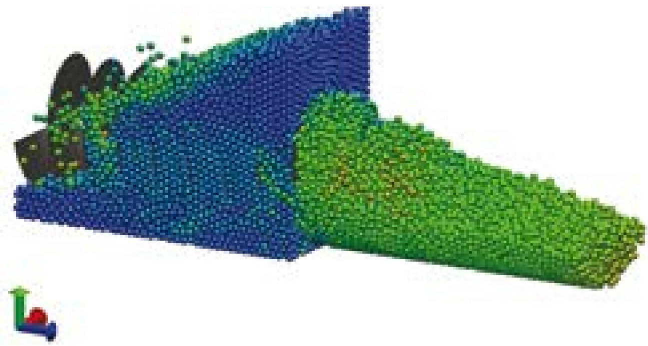


Fig. 7 - Snapshot of the screw feeder simulation coloured by particle velocity. The screw pushes the particles out of the box into the tube. Colours indicate particle speed.

If you are interested in Mercury-DPM?

Hopefully, by now you are interested in trying out Mercury-DPM for yourself. If you would like more information about the code, it can be found on the Mercury-DPM website http://www2.msm.ctw.utwente.nl/MercuryDPM. Alternatively, you can obtain updates and information about the code by joining the mailing list. To do so, simply send an e-mail to listserv@lists.utwente.nl with subject: subscribe and Body: MERCURY-USER <your full name>. This is a low volume mailing list and typically you will receive no more than one e-mail a month. The code itself is available for a public svn repository and details of how to obtain and install it can also be found on the website.

Mercury-DPM was originally started as a research code at the University of Twente, to meet a local need for a tool that was not available in existing simulation codes. Since then it has grown and gained a few tens of external users, until now purely by word of mouth. Therefore, we have decided to make an official public release of the code, which will coincide with the publication of this article.

The Mercury-DPM release philosophy and guarantees to users

Mercury-DPM is an actively developed open-source scientific research tool, which works on a kernel and driver pattern. Some of the authors have used these types of packages before and have often run into the problem to spend time developing the driver, and then a new version of the kernel comes out and nothing works anymore. Then you have to spend time rewriting your driver to get back to a square one.

We are already quite happy with our interfaces in Mercury-DPM and expect them to change very little in the future. However, we will also give the following two guarantees. Any driver code written for version 1.x will work in each version 1 and 2 kernels. New interfaces and modifications to interfaces will initially be introduced in parallel to the old interfaces. The use of an old interface will throw a warning to the users that the interface is to be withdrawn in the next major kernel update and will explain how to convert to the new improved interface. Secondly, there will not be more than one major kernel update per year. This means that any driver code written in the current version of the kernel is guaranteed to work in all new versions for the next two years, at least. Moreover, if after every major kernel update, i.e. once a year, you spend a little time responding to the warning your code generates, it will always work in the future version of the Mercury-DPM kernel.

Mercury-DPM is still actively developed, and we have many grand plans for future features and extensions. These include added smooth particle hydrodynamics, direct coupling with continuum solvers, a graphics interface to aid ease of use, etc. Finally, if Mercury-DPM does not have a certain feature you need, we are always open to collaborate and to add such a feature. Actually, some of our current features arose in exactly this fashion; for example, the helical rotating screw wall.

Code development acknowledgement

Mercury-DPM was started by Anthony Thornton and Thomas Weinhart, and is currently actively developed by Thomas Weinhart, Anthony Thornton and Dinant Krijgsman, with input from Stefan Luding and Onno Bokhove. Stefan Luding provided a complete working particle simulation code with a state-of-the-art set of contact models that has been used as both a validation tool and as a reference guide for various features of Mercury-DPM. Additionally he has provided a great amount of theoretical and technical support in the area of (advanced) contact laws for granular materials and coarse graining. Rudi Fransen developed the current support for visualizing the output of the code using VMD. Ate te Voortwis is currently working on the parallel distribution of Mercury-DEM.

Financial Support

The development of Mercury-DPM has benefited from financial support provided by grants primarily obtained by Stefan Luding and Onno Bokhove. A full list of the grants that have (in part) supported the development is:

(1) the late Institute of Mechanics, Processes and Control, Twente (IMPACT) as part of the research program "Superdispersed multiphase flows";
(2) STW project 11039.
(3) NWO VICI grant 10828;
(4) DFG project SPP1482 B12;
(5) FOM project 07PGM27
(6) STW MuST project 10120.

We would like to thank all parties for the essential financial support they have provided to this project.

Anthony Thornton, Dinant Krijgsman, Rudi Fransen, Sebastian Gonzalez, Deepak Tunuguntla, Ate te Voortwis, Stefan Luding, Onno Bokhove, Thomas Weinhart

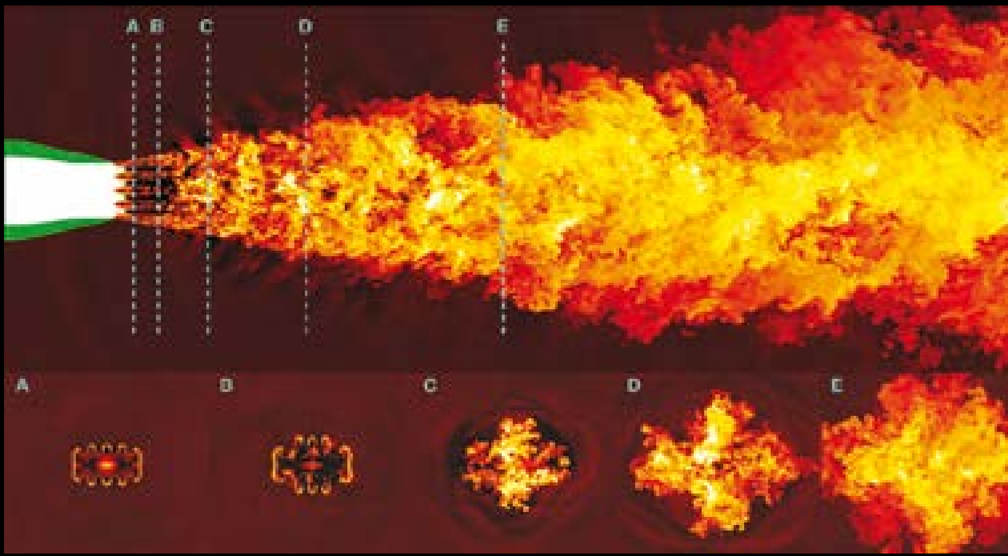
LA DEFORMAZIONE PLASTICA DELL'ACCIAIO: I PRODOTTI LUNGHI

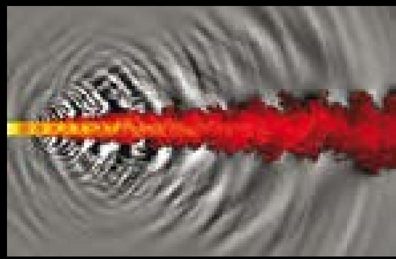
EnginSoft ospiterà a Bergamo il 13 Marzo p.v., presso il proprio Competence Center all'interno del Kilometro Rosso, una tappa del Corso Itinerante organizzato da AIM – Associazione Italiana di Metallurgia, a tema: "La deformazione plastica dell'acciaio: i prodotti Lunghi".

L'iniziativa tecnico-formativa dell'Associazione si articolerà in 6 appuntamenti a partire dal 6 Marzo p.v. ed è strutturata in una parte teorica, sui principi metallurgici di deformazione plastica dell'acciaio, e una pratica che contempla processi e macchinari. Il ruolo degli ingegneri di EnginSoft, specialisti in simulazione virtuale, sarà fondamentale nell'illustrare ai corsisti avanzate soluzioni tecnologiche CAE dedicate alla laminazione dell'acciaio. Marcello Gabrielli e Andrea Pallara, in veste di tutor, presenteranno, con il supporto di AFV Beltrame, con casi concreti, l'approccio al problema di laminazione attraverso la simulazione di questo processo metallurgico al fine di affinare la progettazione dei manufatti così ottenuti ottimizzandone il disegno e il relativo ciclo di produzione.

Per informazioni: Segreteria AIM
Tel.02.76021132; e-mail: info.aim@aimnet.it

ASSOCIAZIONE ITALIANA DI METALLURGIA





CHARLES: high-fidelity compressible flow solver

Leveraging advancements in high-performance computing, Large eddy simulation (**LES**) is emerging as an accurate yet cost-effective computational tool for the prediction of high-speed turbulent flows and their acoustic fields.

CHARLES is Cascade's high-fidelity unstructured compressible flow solver for **LES**, ideally suited for aeroacoustic applications involving high-speed flows and complex geometries. **CHARLES** solves the spatially-filtered compressible Navier-Stokes equations on unstructured grids using a novel finite-volume method where the flux is computed at each control volume face using a blend of a non-dissipative central flux and a dissipative upwind flux. To minimize numerical dissipation and dispersion, the value of the blending parameter is computed locally based on an analysis of the discrete operators such that it is essentially zero in regions where the grid quality is good and the scheme based on the central flux is discretely stable and non-dissipative. In regions of less-than-perfect grid quality, the blending parameter is automatically increased to prevent the pure central scheme from introducing numerical instabilities that would contaminate or destabilize the simulation. Because the local blending is based strictly on an analysis of the discrete operators (and not the solution), the operators can be pre-computed and stored as a pre-processing step at the start of the solution.

Because the underlying numerical method has minimal numerical dissipation, it is critical to employ a sub-grid model to account for the physical effects of the unresolved turbulence on the resolved flow. Several sub-grid scale models are available, including the Dynamic Smagorinsky and Vreman models.

Shocks, like sub-grid scale turbulence, are also sub-grid phenomena and thus require modeling to account for their effect on the resolved flow. However, unlike sub-grid scale turbulence, they are localized in the flow and a surgical introduction of modeling is potentially more appropriate. **CHARLES** uses a hybrid central-ENO scheme to capture shocks, along with the HLLC approximate Riemann solver. The hybrid switch based on the relative smoothness in reconstructed pressure and density is used to activate the 2nd-order ENO shock capturing scheme. This switch ensures that shock-capturing dissipation is only applied local to discontinuous regions, and the bulk of the simulation is based on the low-dissipation and dispersion scheme to produce accurate turbulent transport and mixing.

CHARLES has been used to investigate a wide range of high-speed unsteady flows on complicated adapted grids in both simple and complex geometrical configurations. The space-time databases computed by **CHARLES** form the inputs for other post-processing tools in Cascade's infrastructure, include the far-field noise solver **FWH.**

Top left: Instantaneous temperature field for a hot supersonic jet issued from a rectangular nozzle with chevrons. In collaboration with J. W. Nichols and S. K. Lele, Stanford University. Nozzle geometry courtesy of James Bridges, NASA Glenn research center. The mesh contains approximately 90 M cells and the simulation was performed on 60000 processors (Computational resources provided by Argonne National Labs).

Top right (upper): Instantaneous temperature and pressure field for a supersonic jet impinging on a flat plate. Top right (lower): Instantaneous temperature and pressure field for a heated over-expanded supersonic round jet. (Computational resources provided by ERDC and AFRL supercomputing centers).

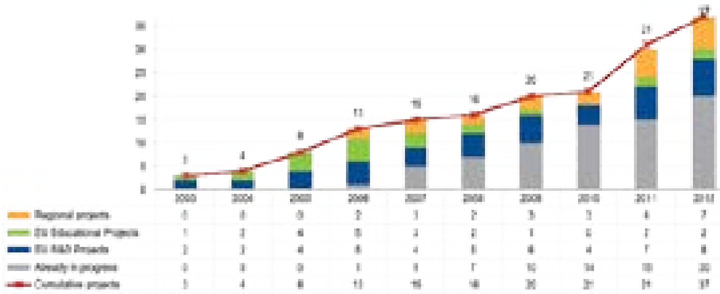
2445 Faber Place, Suite 100, Palo Alto, CA 94303-3347
T. 650.521.0243 F. 650.521.0679 **www.cascadetechnologies.com**

CASCADE TECHNOLOGIES

a partnership with ENGIN SOFT

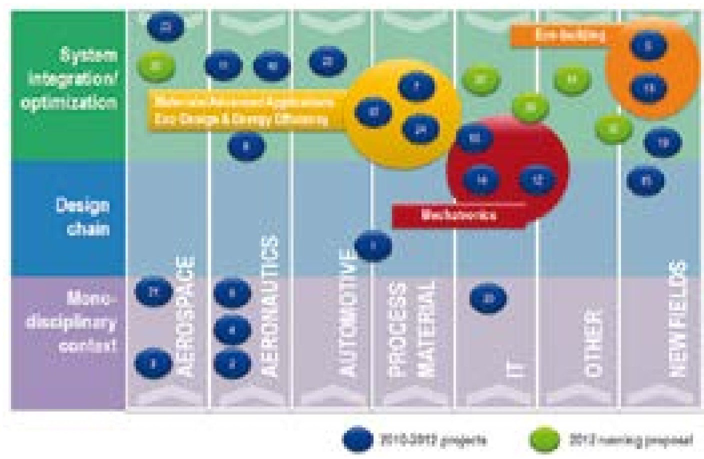
The partner for Research and Technology Transfer

The ever increasing advances in computing and computational power have brought the discipline of Simulation-based Engineering and Sciences (SBES) to the threshold of a new era. What seemed impossible only a few years ago is now possible and the research community in this vast discipline is far from done, there is so much more that is still to be done. This is a very exciting time for engineers and scientists involved in SBES research.



Just think of the challenge of engaging in multi scale-modeling where the conventional methods cannot treat physical phenomena operating across large ranges of time or spatial scales. We need to rethink the ways we conceive SBES methodologies, a challenge that transcends the capabilities of any single scientist or engineer. This is why EnginSoft plays a leading role in many consortium; we understand that as a community of scientists, we can do so much more than as individuals.

In fact, EnginSoft together with its partner network has a long history of collaborative research. We have participated in a wild variety of research projects including many EC projects, Clean-sky, ESA, IMS platforms, SBIR/STTR and many others.



Some of the research projects we have recently participated in include MUSIC (MUlti-layers control&cognitive System to drive metal and plastic production line for Injected Components), BENIMPACT (Building's ENvironmental IMPACT evaluator & optimizer), RLW Navigator (Remote Laser Welding Navigator), NEWAC (NEW Aero Engine Core concepts), MELiSSA (Micro-Ecological Life Support System Alternative), NADIA (the New Automotive components Designed for and manufactured by intelligent processing of light Alloys), and many others…

In addition to participating to research projects, EnginSoft offers financial consulting services for companies engaged in co-funded projects. EnginSoft can help your company to manage and coordinate your research project by:

- Offering support when applying for funding.
- Help you manage and finalize the project.
- Help you select technical experts or laboratories who can provide complementary know-how.
- Providing fast and timely disclosure on grants and low-interest loans available throughout Europe for co-funded research projects.
- Selecting and training research personnel.
- Finding international partners.

For more information:
Angelo Messina, EnginSoft
info@enginsoft.it

VOCAL FAN: Virtual Optimization CFD platform ALlowing Fan noise reduction

The industry demand

During the last years within the aerospace sector (but also in the automotive and home appliance industry, to cite only the most relevant examples) a remarkable and always growing importance has been gained by the need to properly design engine cooling fans, which generally cool cross-flow heat exchangers or similar devices. This has brought to the need of research activities, capable to allow a progress in the understanding of the aerodynamic noise and thus to provide an effective support for the compliance with efficiency constraints imposed both by the market and the national and international regulations (within the automotive sector, to cite only a quite known example, the EURO5 regulations call fan suppliers of engine cooling system for an increase of global efficiency, volume flow rate and pressure raise, as well as a reduction of noise).
Given the aforementioned considerations, the fan is today considered as part of a more complex equipment (electric motor + fan = electric fan), in which efficiency, power consumption and noise are to be targeted at a system level.
Therefore, being efficiency of the electric motors already pretty high and almost at an optimal level, a lot of design work and research is being devoted to the fan blades, thus driving improvements particularly in the aerodynamic design, which is responsible of most of the efficiency and performance of the system in terms of pressure raise, volume flow rate, power consumption and noise generation.

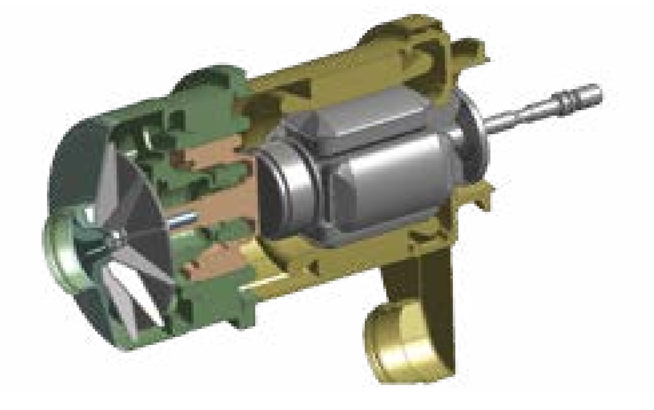
The Consortium

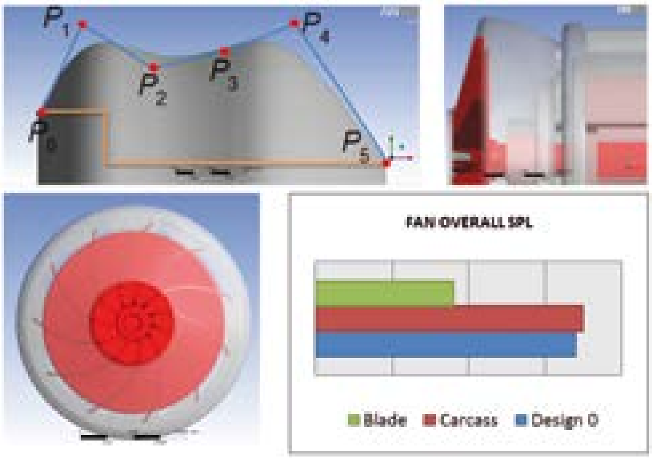
The project, coordinated by EnginSoft, involves one industrial partner from France: Thales Avionics Electrical Systems (TAES).
Thales Avionics is a part of Thales group, a worldwide company involved in different civil and military markets: defense, aerospace, space, security, ground transportation. In particular, the activity of Thales Avionics, based in Chatou (Paris), is focused in Electrical Power Systems in aerospace.

> EnginSoft contributes to the reduction of noise emissions in civil aviation as partner of the European project VOCAL FAN

The problem

A new generation of starter/generator for regional aircraft and business jet has to be developed. The machine is composed by a 3 stages' alternator where the windings of the rotors and stators are cooled by air; the airflow is produced by an integrated fan. Different operating conditions characterize the machine. A first initial design has been supplied by Thales. The objective of the project is to investigate the





noise generated by this original design and then to evaluate different geometric modifications in order to diminish the noise while keeping or improving the other fluid dynamic properties (fan head, volume flow rate, fan efficiency, etc.).

The procedure and the results

The fluid dynamic characterization of the original device has been performed using CFD tools (ANSYS ICEM and ANSYS CFX). After the evaluation of the system performances, the sound pressure level (SPL) has been computed in different internal locations using the spectral analysis. The sound, evaluated for different operating conditions, has been related to the flow behavior showing that the most critical acoustic zone is the region immediately downstream the fan location.
The project has then been developed in two different directions:

- Modification of the external carcass downstream the fan while keeping the original fan;
- Modification of the fan blades while keeping the original carcass.

For all the new configurations designed and analyzed, the main goals were the increase of the fan efficiency and the decrement of the noise level within the machine.
The carcass has been modified by means of a Bezier curve. A 5th order Bezier curve has been used and the control points has been moved to create different geometries. The evaluation of different designs has shown that this kind of modification is not relevant for the performances and for the noise emission. No significant improvement has been identified.
The fan in the original device is characterized by pure radial blades. The modification introduced in the second part of the project is the substitution of radial blades with curved blades (backward or forward). Using different blades' angle it results in different flow behavior and then different noise emission. An optimal design has been individuated using backward blade: an improvement of efficiency of the fan has been found and a huge decrement of noise emission individuated. Such an important results is a combination of two aero-acoustics effects:

- A more efficient fan is characterized by a lower level of noise emission.
- Using this new fan, the original carcass behaves like a resonator and the characteristic frequency of the fan (Blade Passing Frequency) is deleted, resulting in a low sound pressure level.

The role of EnginSoft

VOCAL FAN offers EnginSoft an important opportunity to bring its contribute to make significant advancements in the reduction of noise emissions in civil aviation. EnginSoft worked closely with Thales in the development and optimization of new generation of starter/generator. The experience of the CFD Team of EnginSoft in simulation and design of air cooled systems and the experience in aero-acoustic simulations have been crucial for the success of the project.
EnginSoft has been the coordinator of the entire project.

Acknowledgments

The research leading to these results has received funding from the Clean Sky, the most ambitious aeronautical research program ever launched in Europe. Its mission is to develop breakthrough technologies to significantly increase the environmental performances of airplanes and air transport, resulting in less noisy and more fuel efficient aircraft.

For more information:
Michele Andreoli- EnginSoft
info@enginsoft.it

Lezione sull'ottimizzazione all'Università di Padova

Si è tenuta in Dicembre, a cura del Prof. Meneghetti del Dipartimento di Ingegneria Industriale dell'Università di Padova, una lezione sui temi dell'ottimizzazione. Il nostro ing. Franchini ha illustrato, ad una quarantina di studenti dell'ultimo anno, i concetti principali della disciplina che ad oggi va sotto il nome di MDO (Multi - disciplinary Design Optimization), lo stato dell'arte delle metodologie ad essa afferenti ed alcuni casi applicativi per la risoluzione di problematiche di statica e dinamica strutturale basati su modeFRONTIER in accoppiamento con la piattaforma ANSYS. La lezione fa parte di una serie di iniziative nell'ambito del Programma Accademico di EnginSoft volto a diffondere nelle Università Italiane le nostre tecnologie e know-how per instaurare più stretti rapporti di collaborazione sia formativi che di Ricerca.

Per maggiori informazioni
Lorenzo Benetton - EnginSoft
l.benetton@enginsoft.it

New partnership between NAFEMS and EnginSoft to further develop the Competence Management System

Competence management has emerged over recent years in the corporate world as a central theme between technical management, project and program management, and HR management. It enables companies to hire, develop and retain the best human capital and provides solid foundations to cope with the rapid change of the technical knowledge and the development of the job market.
Over the last year, NAFEMS and EnginSoft have partnered in order to further develop a set of competence management tools dedicated to the Analysis and Simulation industry, originally developed in the frame of the EASIT2 project, a Leonardo da Vinci project co-funded by the European Community under the People program.
NAFEMS is the not-for-profit International Association of the Engineering Modeling, Analysis and Simulation Community, aiming at improving the professional status of all persons engaged in the use of engineering simulation, establishing best practices and continuously improving the education and training in the use of simulation techniques.

Product of this joint effort is the Competence Management System, software developed and supported by EnginSoft, that enables companies and individuals to identify, track and manage Analysis and Simulation competencies.
The system is built around the NAFEMS Educational Base, an extensive database of “standard” competencies and related educational resources, covering several areas of competence, such as Finite Element Analysis, Computational Fluid Dynamics, Materials for Analysis and Simulation, Fatigue, Dynamics and Vibration, Optimization, Simulation Data Management, Composite Structures and Materials, and many more.

Each competence that is described in the database is linked to appropriate educational resources, which include books, papers, articles and codes of practice. Overall, the Competence Management System references more than 1000 educational resources. Among these, the resources published by NAFEMS are directly linked to the new NAFEMS Corporate e-Library service, a new online service dedicated to the NAFEMS corporate members. The NAFEMS Corporate e-Library puts the full and extensive library of educational books and resources published by NAFEMS over the last 30 years at the fingertips of the Competence Management System user. The NAFEMS Corporate e-Library provides instant access to over 140 NAFEMS publications, including the newest releases, and it also provides convenient access to the most recent articles from the NAFEMS Benchmark magazine. The Competence Management System leverages the Educational Base, extending its scope by

Fig. 1 - The new NAFEMS Corporate e-Library system provides to NAFEMS Corporate Members direct access to an electronic version to a wealth of NAFEMS publications and articles

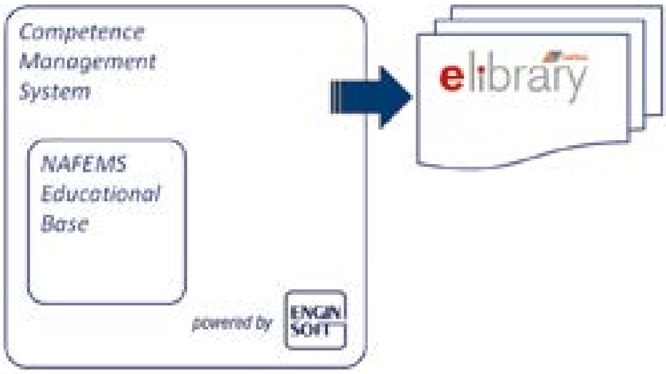


Fig. 2 - The Competence Management System references more than 1000 books, articles, papers and codes of practice, some of which directly available through the NAFEMS Corporate e-Library service

adding useful features such as the ability for companies to fully customize the competence database. Companies can add their CAE system-specific or industry-specific competencies to the database, and, crucially, they can group the existing competencies in order to define subsets that are meaningful to their specific industry and processes. Furthermore, competencies and subsets can be assembled in order to create "blueprint" competence profiles for employees, useful for example to design and streamline the training path of the technical staff. The system can also be integrated with the current competence assessment workflow inside companies, such as an annual review and the approval process from line managers and/or HR managers.
Group management features combined with the reporting features of the system can be used to enable effective competence management not only for individuals, but also for work groups and projects teams: this will help companies to leverage the sound management of competencies provided by the Competence Management System to assess competency-related project risks and consequently improve project planning.

The Competence Management System is a secure web application, and can thus be accessed directly over the web: an online standard version of the Competence Management System will be provided to NAFEMS members free of charge. The system can also be installed inside a company Intranet for additional security and confidentiality: in the latter case, it can be interfaced with existing applications such as ERPs or HR management systems allowing for additional customization.

For more information:
Giovanni Borzi - EnginSoft
info@enginsoft.it

Atkins Water Commits to Saving Time and Money

EnginSoft UK worked with Innovyze, the developers of widely used water industry software InfoWorks CS over the past three years to develop an "interface" to automatic control and exchange data between modeFRONTIER and InfoWorks.

We continually work directly with industry users through a Steering Group Committee to apply our expertise in this sector. A number of case studies by early adopters in the committee have been completed, realising the benefits of time efficiencies and costs saved. Atkins has made the decision to take advantage of modeFRONTIER in 2013.

Through connecting InfoWorks CS to modeFRONTIER using the interface users are able to benefit from automatic calibration of InfoWorks CS models resulting in a significant time reduction for model verification, automate the optioneering process to develop a range of solutions that provides more options to the decision maker, users will also gain improvements on solution quality by automating the extensive manual checking process also leading to a dramatic increase in utilization of the InfoWorks CS license allowing resources to run after hours including throughout holidays, weekends and overnight.

The recent release of the Graphical User Interface version 3.0 allows for an advanced user friendly experience to further benefit and exploits the capabilities of the interface. More efficient capital costs solutions are developed with the user being able to understand the trade-offs between costs versus quality/confidence (the "Pareto" solutions). Highlights of the Graphical User Interface include an Interactive map and chart for target settings, with easy to set up cost model and DG5 flood monitoring.

For more information:
Bipin Patel - EnginSoft UK
info@enginsoft.com

ATKINS

La FORMAZIONE CFD in EnginSoft

EnginSoft è impegnata in attività di formazione fin dai suoi esordi. Tra le iniziative in tale ambito, si annovera la partnership con il consorzio TCN, associazione il cui fine è la divulgazione tecnico-scientifica nel mondo del CAE. Da anni inoltre EnginSoft organizza e promuove direttamente corsi, sia per quanto concerne la formazione di base che specialistica, all'utilizzo dei software CFD nei più disparati settori industriali.
È all'interno di questo naturale e importante ramo del suo mandato, che l'azienda ha sviluppato competenze e coltivato relazioni che le consentono di giocare un ruolo da protagonista, e che trova una sua tangibile testimonianza nell'essere un ente riconosciuto dal Ministero dell'Istruzione, dell'Università e della Ricerca (MIUR).
Tutto ciò, tuttavia, non può che essere un punto di partenza. L'aumento vertiginoso dell'utilizzo dei software CAE degli ultimi anni, infatti, è al tempo stesso causa ed effetto del corrispondente significativo aumento delle capacità previsionali della realtà da parte di questi applicativi che, con esistenti e nuovi modelli fisici e numerici, consentono lo studio di fenomeni, nemmeno lontanamente pensabile solo fino a pochi anni fa. La corrispondente espansione delle capacità dell'universo informatico ed elettronico, su cui questi applicativi si poggiano, fa il resto.
Accade quindi che, in EnginSoft, una volta consolidate le competenze relative a tematiche che si rendono via via gestibili ed affrontabili con gli strumenti della simulazione numerica e di sicuro interesse del mercato, queste diventino parte integrante dell'offerta formativa di EnginSoft verso le aziende che ne possono beneficiare.
È il caso, fra diversi altri, di alcuni corsi avanzati di CFD, la cui introduzione è stata agevolata anche grazie alle competenze acquisite da EnginSoft in seguito alla partecipazioni a progetti europei. Tali corsi sono:

- **Corso Avanzato di Aeroacustica**: i settori cui questa tematica può trovare applicazione sono i più svariati, dall'impiantistica (fan, blower, valvole, …), agli studi di comfort, all'edilizia ambientale, …
- **Corso di Interazione Fluido-Struttura**: anche in questo caso i campi di applicazione sono molteplici e consentono di affrontare un problema ingegneristico con una visione più ampia, passando da un approccio mono-discplinare sul componente singolo ad uno multi-fisico sul un sistema, inteso come aggregazione di più componenti che si influenzano a vicenda.

Di seguito, una breve descrizione tecnica dei contenuti di questa due offerte formative.

Per ulteriori informazioni:
http://www.enginsoft.it/formazione
Luca Brugali, EnginSoft
info@enginsoft.it

ANSYS CFD:
Corso Avanzato di Aeroacustica

L'aero-acustica è quella parte dell'acustica dove il suono viene generato aerodinamicamente da un fluido, in particolare aria, e tale generazione può avvenire in diversi modi: campo libero senza superfici solidi (suono generato dalla turbolenza, es. jet noise), campo libero con superfici solide (fan noise, rumore di cavità), campo confinato (rumore nei condotti, silenziatori)
Il corso, della durata di un giorno, fornisce una panoramica completa dell'utilizzo dei software ANSYS CFD (Fluent e CFX) per impostare e risolvere un'analisi aeroacustica.. Dopo una prima introduzione generale sull'acustica per definire e richiamare i concetti base, vengono affrontati i diversi metodi disponibili per la simulazione CFD del rumore. Vengono investigati metodi complessi di simulazione diretta del rumore ma anche metodi più semplici per una stima del rumore da utilizzare in fase di predesign.
Affiancata ala parte teorica si trovano alcuni esempi ed esercitazioni, tratte direttamente dall'esperienza pratica su base industriale.
Il corso si propone di fornire ai partecipanti una maggior conoscenza nel campo dell'aero-acustica sia per quel che riguarda la modellazione sia per quel che riguarda l'utilizzo pratico degli strumenti CFD.
I destinatari del corso possono essere ingegneri e progettisti che avvertono l'esigenza di utilizzare un software CFD (ANSYS-CFX o ANSYS FLUENT) per lo studio dell'aeroacustica nel contesto della propria attività e che affrontano per la prima volta problemi di modellazione numerica in ambito acustico.

ANSYS CFX-MECHANICAL:
Corso Interazione Fluido-Struttura

Il corso è della durata di 2 giorni e si propone di fornire le linee guida e operative per una corretta impostazione di problemi di interazione fluido-struttura (Fluid Structure Interaction, o FSI) all'interno della piattaforma ANSYS Workbench.
In particolare, l'impostazione metodologica di tali problematiche verrà affrontata considerando i pacchetti software ANSYS Mechanical (per la trattazione della meccanica) e ANSYS CFX (per la trattazione della fluidodinamica), proprio in ragione della loro semplice integrazione all'interno dell'ambiente ANSYS Workbench.
Il modo di interagire dei due sistemi software verrà illustrato in relazione ad applicazioni con presenza di carichi fluidodinamici in direzione normale (pressione) e tangenziale e/o di carichi termici.
Durante le due giornate verrà descritto, anche mediante semplici esempi pratici, come impostare problemi FSI sia di tipo 1-way, dove i carichi (le deformazioni) sono tali da non influire significativamente sulle deformazioni (sui carichi), sia di tipo 2-way, dove carichi e deformazioni sono più o meno fortemente accoppiati, richiedendo quindi delle iterazioni tra i nodi CFD e FEA. Il corso prevede altresì di fornire linee guida per la discretizzazione (meshatura) dei modelli interessati all'interazione, la cui griglia di calcolo dovrà opportunamente essere deformata (FSI 2-way).
Oltre a dare indicazioni di carattere operativo, verranno forniti dei cenni al procedimento di trasferimento dei file (ed il corrispondente trasporto di grandezze fisiche) tra i modelli utilizzati nelle due fisiche coinvolte.
È preferibile che vi sia da parte dell'utente una conoscenza, seppur minima, degli applicativi coinvolti, ovvero le modalità operative all'interno dell'ambiente Workbench, il pacchetto per l'analisi fluidodinamica 3D ANSYS CFD ed il software per l'analisi FEM ANSYS Mechanical.
I destinatari del corso possono essere ingegneri e progettisti, sia fluidodinamici che strutturisti, i quali abbiano già una conoscenza, almeno di base, dei codici coinvolti.

ANSYS Italian User Meeting
Salsomaggiore Terme 20-21 Giugno 2013

Si terrà a Salsomaggiore Terme- Parma - la conferenza riservata agli utenti italiani di prodotti e soluzioni fondate sulla piattaforma ANSYS. Lite motive dell'incontro è "Convergenza" declinata su due direzioni. La prima è relativa al trend Convergente tra le differenti tecnologie che compongono la suite ANSYS ovvero la crescente necessità, espressa dagli utenti, di accedere a nuove funzionalità in ambienti multi-fisici sempre più integrati e performanti. La seconda si riferisce alla necessità di condividere, incontrarsi, scambiarsi esperienze tra addetti ai lavori. E questa motivazione, più che mai in questo momento storico per il sistema-paese Italia, è uno sprone a unire le forze per uscire da un empasse che da troppo tempo a questa parte sta bloccando la nostra economia.

La due giorni di Salsomaggiore vedrà EnginSoft ed ANSYS Italia co-protagonisti nel presentare, in anteprima italiana, la R.15 di ANSYS il cui rilascio per gli utenti è previsto a fine 2013. Ad illustrare le novità di prodotto e le direzioni di sviluppo saranno alcuni VIP di ANSYS Corporation e tra questi Swaminathan Subbiah – Vice Presidente e responsabile di prodotto e delle strategie di marketing e Amar Bouali – Vice presidente e responsabile delle tecnologie Esterel.
L'evento si articolerà attraverso una sessione plenaria, nella quale verranno presentate le direzioni del prossimo sviluppo e avverrà la presentazione del concorso per i migliori Paper, e a seguire sessioni

parallele dedicate alle singole tecnologie. In contemporanea si terrà l'High Frequency Simulation Conference dedicata ai prodotti per la simulazione dei fenomeni elettromagnetici di antenne e dei circuiti elettronici.

Ma la promozione della R.15 di ANSYS non si fermerà a Salsomaggiore Terme. L'International CAE Conference – www.caeconference.com, in programma il 21 e 22 Ottobre, consoliderà il lancio del nuovo ambiente ANSYS con importanti novità applicative e case study dai beta tester.

Nell'area tecnica, ricavata all'interno dello spazio espositivo, clienti vecchi e nuovi avranno l'opportunità di incontrare gli ingegneri e Product Manager EnginSoft disponibili per demo, presentazioni e per rispondere alle necessità tecniche.

Come accennato, la Conferenza sarà il trampolino di lancio dell'ANSYS Best Paper Award 2013 che premierà, nel corso della Conferenza, i talenti italiani attraverso lavori (paper) provenienti da due categorie distinte: Industria e Ricerca. La prima è riservata alle Aziende e la seconda ai Centri di Ricerca e alle Università (utilizzatori di licenze teaching, research o associate).
Maggiori dettagli sul concorso e le modalità di partecipazione su: www.amiando.com/ansysugm2013.htm

Arrivederci a Salsomaggiore Terme!

EVENT CALENDAR

March 12-14, 2013
Paris - France
JEC Europe Composites Show & Conferences
http://www.jeccomposites.com/events/jec-europe-2013

March 20-22, 2013
Ravenna - Italy
Offshore Mediterranean Conference & Exhibition
www.omc.it/2013

June 9-12, 2013
Salzburg - Austria
NAFEMS World Congress
www.nafems.org/congress/

June 19-21, 2013
Mannheim - Germany
ANSYS Conference & 31. CADFEM Users' Meeting
www.usersmeeting.com

June 20-21, 2013
Salsomaggiore Terme (Parma) - Italy
ANSYS User Group Meeting Italia 2013
www.amiando.com/ansysugm2013.html

October 21-22, 2013
Northern Italy
International CAE Conference 2013
www.caeconference.com

2013 SEMINARS
Stay tuned to www.enginsoft.it/eventi
for the complete program of seminars in 2013

FREE Newsletter Printed Copy

Do you want to receive a free printed copy of the EnginSoft Newsletter? Just scan the QRcode or connect to: www.enginsoft.it/nl

The 9th International Rolling Conference and the 6th European Rolling Conference will be jointly organized by AIM, the Italian Society for Metallurgy. Thanks to a long and successful series, the Rolling conferences have become the premier international event for rolling practitioners worldwide. The target audience embraces the whole metals rolling community: flat and long products rolling, hot and cold rolling and ferrous and non-ferrous rolling. The forum will cover product and process topics, encompassing rolled product properties, quality and applications, in addition to the design, control and management of mill assets. The Conference will bring together rolling practitioners, steel producers, plant designers and researchers and it will provide a forum for best practices and state-of-the-art technology. It will also take a look at developments in the foreseeable future.

TECHNICAL FOCUS

The Conference will cover topics concerning:

- New process, technology and facilities for flat products, long products and pipes
- Thin slab casting and inline rolling
- Strip casting
- Thermomechanical processes
- Automation, measurements and control technology
- Re-heating furnaces
- Galvannealing of rolled products
- Equipment and maintenance
- Mathematical modeling and simulation
- Surface modification and steel deep-working
- Clean rolling
- Advanced products development and application
- Health & Safety

Conference website: www.aimnet.it/rolling2013.htm

EnginSoft, together with Transvalor will present two papers:

1. Complete simulation and optimization of the production process of a bearing ring in 100Cr6 steel
   A. Sartori - Muraro, Zermeghedo, Italy
   M. Gabrielli - Enginsoft, Italy
2. A complete 3D simulation tool for sequence of continuous casting and rolling including perfect transfer of data in between operations
   O. Jaouen - Transvalor S.A., France

Aluminum Two Thousand
14 – 18 May 2013 Milano

Aluminium Two Thousand Congress is an international not-to-be-missed conference about aluminum. After 7 editions, it has become a very important and irreplaceable meeting point for all the operators working in the Aluminum industry.
Highly qualified "Aluspecialist" from companies, universities and associations come from all over the world to present to a very specialized and international audience the most innovative technologies and applications in the aluminium field. Last year in Bologna, over 350 experts from 43 different countries attended the conferences.

What is the goal of the Congress?

To analyse all the aspects of the Aluminium chain, to meet colleagues from around the world, to exchange ideas and advice and to give your own contribution to the growth of the aluminium industry and the improvement of technology. Customers and suppliers can improve their business and become stronger as an "aluminium community"

EnginSoft Partecipation

Title: ROBUST DESIGN OPTIMIZATION OF HIGH PRESSURE DIE TO PRODUCE PISTON RODS
In general, automotive component design originates from structural FEM simulations and mechanical behavior verifications, it is a geometry input for the manufacturer. The mechanical properties of HPDC components can be affected by gas porosity, shrinkage and other different defects as well as oxides or cold shots. This means that the HPDC is an attractive process to achieve high levels of productivity, yet its set up and the optimization of the die design are very complex tasks.
The problem can be divided in sub-tasks:

1. Traditional structural analysis of the designed piston rod to verify which are the critical zones, where are the maximum compressive and tensile stresses, and to evaluate the stress level compared to the ultimate tensile strength of the material;
2. Optimization of the production process (take advantage of Automatic Optimization technologies) for defining the dies and process parameters suitable for the production of Piston Rod maximizing its quality;
3. Advanced structural analysis by exploiting the mechanical properties of the cast from the process analysis, to perform the final structural verification and to validate the production process.

The presentation of EnginSoft aims to show the innovative techniques for the design using the integration of virtual simulation tools to optimize the quality of the finished product, indicating also the basics.

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

21-22 October 2013

www.caeconference.com