Accelerating the development of general purpose engines using modeFRONTIER

On the frontier of drilling technologies

The simulation as competitive advantage for a company producing air filtration units

Morphology of membrane formation during solvent evaporation

Tolerance analysis on Emax 2 low voltage air circuit-breakers series

Multi-Objective Optimization of XBA Sentinel Antenna

Ship Innovation is: thinking composite
I am delighted to announce that the Call for Papers is now open for this year’s International CAE Conference! As we wait in anticipation for this exciting line-up of papers, from an ever growing network of organisations that cross all industry sectors, we are delighted to share with you the ongoing advances being made in simulation technology within EnginSoft and its customer network. Each article shows how the application of simulation technology is being used to improve product performance and foster the spirit of innovation around the world.

In this edition we will be sharing insights into the principal role that simulation is playing within leading companies. Honda shares their aim to accelerate development (Page 8), the CEO of Verme Projects explains their goal in maintaining innovation (Page 6), and TAMA Air Filtration demonstrate the importance of simulation in enhancing competitive advantage (Page 11). The articles highlight how modern engineering companies are committed to new areas of simulation practice in order to achieve their goals.

ABB, a global leader in power and automation technology is no exception. On page 16, we learn how simulation is utilised in the design and development of a high capacity switchgear with a 45% reduction in plan area. In partnership with EnginSoft they were also able to meet all the design objectives, be able to reduce the time to market and benefit from scrap and rework savings.

With many new simulation practices, computational resource can often become stretched and require good simulation process management. Benelli demonstrates how the implementation of modeFRONTIER has enabled them to overcome this challenge by maximising their licence and hardware resources. This has been able to reduce their time to market from 4 months to 1 month. Read the article on page 31 to find out more.

As simulation continues to flourish with ever new techniques and approaches, we are committed to inspiring industry to greater heights by sharing our common challenges and highlighting the new solutions being used in our network of companies. Together, we are better able to exceed current product performance, meet our own design objectives and deliver value driven solutions to our customers.

Stefano Odorizzi, Editor in chief

For many industries today, engineering simulation is the most powerful tool used in product development. It is unparalleled in the speed in which designs can be understood and each design requirement met. This fact is no longer in doubt.

Slow and high cost developmental testing has in the large part been replaced by simulation. However, why stop there? Is there an even better method? What is the current ‘game changer’ in engineering? These questions are answered by a tool that allows the engineer to standing back from the problem, explore the whole design space through appropriate data mining tools, while an automated process considers all the relationships between each design variable and drives the design towards its optimum. This tool is modeFRONTIER.

A special edition of the Newsletter, dedicated to modeFRONTIER, highlights how it has impacted many industries in the way they do their design and development; indeed you will be amazed by how many familiar everyday products have benefitted from this approach and are featured in this edition – please enjoy and become part of the growing community of business leaders that are reaping the rewards in an increasingly competitive world market.

DOWNLOAD your free copy on: www.enginsoft.com/mlf-newsletter
Contents

INTERVIEW
6 Innovation is “thinking composite”, Massimo Verme shares his vision with us
8 Accelerating the development of general purpose engines using modeFRONTIER
11 The simulation as competitive advantage for a company producing air filtration units

CASE HISTORIES
14 On the frontier of drilling technologies
16 Tolerance analysis on Emax 2 low voltage air circuit-breakers series
21 Implementation of surrogate models to predict the morphology of membrane formation during solvent evaporation
24 Multi-Objective Optimization of XBA Sentinel Antenna
28 Magnetostatic Analysis on ITER Test Blanket Modules
31 Structural optimization of the Benelli Progressive Comfort - Dimensional and weigh optimization under maximum cartridge load
34 Coffee Makers and Homecare Products with a Combination of style and performance
36 Optimization of the production process of a die-casting component for automotive using MAGMAfrontier
40 Structural analysis of Microchannel Condenser Heat Exchangers using Finite Element Techniques
43 FE modeling of a composite sandwich laminate with LS-DYNA for Aerospace applications
49 Uncertainty in simulations: Polynomial Chaos Expansion

SOFTWARE UPDATE
51 Advanced Python Scripting in Kraken
52 Well Placement Optimization Using Genetic Algorithms
54 Multi-Body Dynamics for ANSYS
57 MAGMA 5.3 Fully Optimize

IN-DEPTH STUDIES
59 What is System Level Thermo-Fluid Analysis?
66 MUSIC Project at CO.SUMMIT 2015

RESEARCH
64 Growing food for space exploration: EnginSoft is now part of the EDEN ISS project
66 MUSIC Project at CO.SUMMIT 2015 last 10/11 March 2015 in Berlin

OUR ACKNOWLEDGEMENT AND THANKS TO ALL THE COMPANIES, UNIVERSITIES AND RESEARCH CENTRES THAT HAVE CONTRIBUTED TO THIS ISSUE OF OUR NEWSLETTER
Innovation is “thinking composite”, Massimo Verme shares his vision with us

Eng. Massimo Verme is the CEO of “Verme Projects”, a design company operating in the marine industry. He started in 1996 as a freelance engineer for a famous yacht designer and has become a service supplier to shipyards and ship owners over the past five years. Verme Projects was founded in 2000, and since its beginning, they have had the ambitious idea of improving engineering design by running collaborative projects that reduce development times, yet increase product quality. Today, Verme projects is in collaboration with preeminent designers and boatyards in the industry, to design luxury boats by applying its own hull technology to achieve stunning results.

What do Composite materials represent for an Engineer or a Designer?

The use of composite materials in the marine industry, as well as many other many other fields, has removed many of the design constraints, which until recently, have restricted the designer’s creativity. For those who believe in integrating beautiful aesthetics with sound engineering, ‘thinking composite’ on a daily basis is imperative.

The design is a complex process, how do you organize the different stages of development to guarantee maximum efficiency?

Once the project is outlined on a macro scale, great attention is required in developing the detailed aspects on the micro scale to ensure it is done in an efficient way. For example, the use of numerical tools plays a key role in providing an immediate response to many emerging requirements, such as defining a yacht’s setup. The design is developed through different phases, starting with the styling, to design analysis and finally through to detailed optimization. The detailed phase is guided by the ability to quickly find optimal solutions, by going from a simple handmade draft to deeply detailed FEM models. Even in this second stage it is not possible to ignore shapes and style, as their integration is our aim.

What are the main advantages of integrating CAE tools when designing a boat?

The numerical tools are essential means to completing a project in an efficient way; in the case of composite structures, the value of such integrated tools becomes even higher as they allow us to take a materials’ orthotropic properties and technological processes into account. Verme Projects carries out this kind of activity every day, using ESAComp and ANSYS, which allows us to model complex systems in a very accurate and complementary way. It also allows us to improve the boat’s structural performance by analyzing many different configurations.

The material definition is an essential part of composite design: what are the benefits of using a code like ESAComp, specifically when at the preliminary analysis stage?

With regards to preliminary design, ESAComp allows us to assess the conceptual design of the laminates in an effective way. Each potential composite design can be identified immediately for sandwich panels and local reinforcements. Only later can the material design phase be combined with a FE code for detailed analysis; in this respect the integration with ANSYS gives us the best solution.

The integration between ESAComp and ANSYS offers unquestionable advantages: what examples do you have where you have fully exploited the potential?

A recent case for us was the acoustic optimization of a displacing yacht. In this kind of boat, the acoustic comfort is very important and it is necessary for the structural design to make a tradeoff between weight and strength. Most of the hull’s primary structures were made of sandwich laminates and had to be dynamically evaluated according to the acoustic requirements (e.g. engines and propulsion) on board. Through the features available in ESAComp, a preliminary assessment was performed to study the natural frequency of the main panels, identifying the geometric and laminate configurations which were critically close to the standard cruise speed.

Once the main structural parameters were identified, a model was built in ANSYS to understand the global dynamic behavior of the boat close to the engine supports. The numerical model was then validated through experimental tests on board, reproducing the same operating conditions as in the ANSYS model. A good correlation between numerical and experimental results was obtained in the frequency range of 25 and 75 Hz. Once the structural model was validated, we could then complete a fully configured simulation with a floating boat, gathering the proper considerations and optimizations.

Regarding the implementation of numerical tools and our design procedures, the partnership with EnginSoft has always represented a key element to our success, as it helps us to choose and efficiently implement ESAComp and ANSYS. These solutions, together with the technical support given by EnginSoft help us to remain at the cutting-edge of composites design.

Image at the top: Amer Cento Assembly

Ask the expert in Composites: Fabio Rossetti - EnginSoft
frossetti@enginsoft.it
Accelerating the development of general purpose engines using modeFRONTIER
Power Products R&D center, Honda R&D Co., Ltd.

Mr. Gaku Naoe, Mr. Toshiro Kiura, and Mr. Masami Okubo from the Honda R&D Company Power Products R&D Center discuss how modeFRONTIER has changed the way company approaches the product development process and shed light on how modeFRONTIER’s ease-of-use and exceptional performance has made it the automotive giant’s tool of choice.

Honda R&D Company develops many different types of products. How do you use CAE in the R&D center to develop these products?
Mr. Kiura: Our use of CAE depends on whether it is for technical development or product development. In the first case, we use CAE for the evaluation of new technologies and to verify the possible applications. The opposite occurs in product development, where CAE is used for predicting product performance before making a prototype as it narrows down the specifications and increases the quality of test models.

Mr. Okubo: Unlike Naoe and Kiura, I work in the dedicated CAE group and support product development projects in general. I also have a role in developing simulation technologies.

One of the things we always ask our users is whether they have been successful in reducing time of development projects. Have you been able to shorten project times? How do you proceed with your projects?
Mr. Naoe: Our development times are definitely shorter than they once were. As a result, we’ve had to improve the accuracy of performance prediction before making prototypes. Therefore, the CAE process during the design stage is more and more important to achieving this goal.

Honda R&D Co., Ltd.

CAE is now an integral part of project design at Honda. When and why did you start thinking about using CAE?
Mr. Naoe: I’m in charge of sound, vibration and strength testing of general purpose engines. At university I gained experience in structural analysis and started using CAE for checking the correlation between test results and simple CAE results.

Mr. Kiura: I joined the company with the aim of working in CFD simulation. Because fluid dynamics phenomena are difficult to see, CAE plays a very important part in the understanding and clarification of that phenomena. These days, we use CAE in the development of low-emission engines, improving overall engine technology and in research towards future projects.

Back in 2007 you started using modeFRONTIER. Can you tell us the motivation behind your decision?
Mr. Naoe: We were having problems with sound and vibration on the link design of “EXlink,” the extended expansion linkage engine. At the same time, my colleagues were having problems with balance cooling performance and noise in the centrifugal fan. Together, we decided to give “optimization” a try. Our experience showed us that multi-objective optimization was fundamental in solving the problems and that’s when we came across modeFRONTIER.

What was your first impression of modeFRONTIER?
Mr. Naoe: The first time I used it I thought it was a nice software tool. It was surprising because it was very easy to use - I was able to use it after only half a day’s training. My main task is testing and I often step away from simulation noise in the centroid area. Together, we decided to give “optimization” a try. Our experience showed us that multi-objective optimization was fundamental in solving the problems and that’s when we came across modeFRONTIER.

What was the first project you worked on with modeFRONTIER?
Mr. Naoe: The first time I used it I thought it was a nice software tool. It was surprising because it was very easy to use - I was able to use it after only half a day’s training. My main task is testing and I often step away from simulation noise in the centroid area. Together, we decided to give “optimization” a try. Our experience showed us that multi-objective optimization was fundamental in solving the problems and that’s when we came across modeFRONTIER.

What was it like before the introduction of automated optimization and simulation software?
Mr. Naoe: In the beginning, “Optimization” was often misunderstood. Everyone thought they would get an answer simply by entering some numbers. So it took some time to get for our engineers to accept this new technology. As I already mentioned, we proceed with developments on a project basis. Over the years, personal responsibilities and workloads have become heavier meaning that engineers are required to develop their skills constantly. In order to let engineers know the meaning and necessity of multi-objective optimization, we carried out an educational campaign to spread this technology by showing actual case studies in our company. In the campaign, we insisted that the person in charge had to make the final decision and that it is important that he/she evaluate the physical phenomenon from various angles, by showing multi-dimensional analysis charts, bubble charts, and correlation charts prepared in modeFRONTIER post-processing. After that, because we had achieved results by formulating models and using post-processing capabilities, the number of users increased - modeFRONTIER’s easy to use environment also played a key role in engineers adopting it as their tool of choice. In my case, being in charge of testing, it’s very effective for data mining and targeting test results and not just for its integration with other CAE software. Recently we have been using its clustering capabilities.

Have you ever used modeFRONTIER for purposes other than those you’ve already spoken about?
Mr. Kiura: While we had tried numerical simulation in the CAE group, we had also considered applying optimization to measurement and control. As we need to collect a large amount of data for many purposes in engine measurement, model-based measurement together with multi-objective optimization is very efficient. Especially in the case when there are many control factors, choosing principle parameters and understanding the phenomenon equality as important.

To promote the use of CAE, have you ever used modeFRONTIER in non-technical projects?
Mr. Okubo: Unlike Naoe and Kiura, I work in the dedicated CAE group and support project design at Honda. CAE is now an integral part of project design at Honda.

How do you use CAE in the R&D center to develop these projects?
Mr. Naoe: Our use of CAE is based on the company’s goal of improving overall engine technology and in research towards future projects.

Have you ever used modeFRONTIER for purposes other than those you’ve already spoken about?
Mr. Kiura: While we had tried numerical simulation in the CAE group, we had also considered applying optimization to measurement and control. As we need to collect a large amount of data for many purposes in engine measurement, model-based measurement together with multi-objective optimization is very efficient. Especially in the case when there are many control factors, choosing principle parameters and understanding the phenomenon equality as important.

Optimization is very efficient. Especially in the case when there are many control factors, choosing principle parameters and understanding the phenomenon equality as important.

In order to let engineers know the meaning and necessity of multi-objective optimization, we carried out an educational campaign to spread this technology by showing actual case studies in our company. In the campaign, we insisted that the person in charge had to make the final decision and that it is important that he/she evaluate the physical phenomenon from various angles, by showing multi-dimensional analysis charts, bubble charts, and correlation charts prepared in modeFRONTIER post-processing. After that, because we had achieved results by formulating models and using post-processing capabilities, the number of users increased - modeFRONTIER’s easy to use environment also played a key role in engineers adopting it as their tool of choice. In my case, being in charge of testing, it’s very effective for data mining and targeting test results and not just for its integration with other CAE software. Recently we have been using its clustering capabilities.

Have you ever used modeFRONTIER for purposes other than those you’ve already spoken about?
Mr. Kiura: While we had tried numerical simulation in the CAE group, we had also considered applying optimization to measurement and control. As we need to collect a large amount of data for many purposes in engine measurement, model-based measurement together with multi-objective optimization is very efficient. Especially in the case when there are many control factors, choosing principle parameters and understanding the phenomenon equality as important.

Optimization is very efficient. Especially in the case when there are many control factors, choosing principle parameters and understanding the phenomenon equality as important.
Please tell us why you use modeFRONTIER in combination with systems for engine measurement and applied fields.

Mr. Kiura: The dedicated auto driving system integrated with dynamometer makes advanced measurement possible. However the DOE, optimization algorithm and result analysis capabilities in such systems is limited. Therefore, to enhance the degree of freedom and expandability of the measurement and multi-objective optimization capability, we combined the multi-objective optimization of modeFRONTIER and the auto measurement of LabVIEW. Moreover, there were other purposes for visualizing measurement results for different parameters from various perspectives by using response surfaces and correlation charts, examining the results that designers usually trade off, and establishing the knowledge for targeted designs. Not only that, in more general cases, we use modeFRONTIER to evaluate our own assumptions for the phenomena in which principle factors are not specialized for multi-objective optimization and formulation is not easy. This means that, using modeFRONTIER, we analyze from various approaches and consider the results. modeFRONTIER is our partner who helps us solve the problem.

From your comment, I understand that you use modeFRONTIER for each step of model formulation, parameter DOE decision, auto execution, and post-processing. What is the outcome when using modeFRONTIER?

Mr. Naase: The tests and CAE results that we’ve obtained so far have given us design space information and we used that data to change some conditions. Now, by using multi-objective optimization, we can predict to some extent the underlying phenomena and establish an appropriate strategy. modeFRONTIER gives us a large amount of data and we continue our studies based on that data. For example, the number of principal design parameters was traditionally about 4 for the engine mechanism. In contrast, the number of principal design parameters for EXlink increased to 11 and we were at a loss to know that the parameter studies would be around 400 million designs. We ran about 1,000 to 2,000 calculations with a modelFRONTIER genetic algorithm and found the way to maximize performance. modelFRONTIER enabled us to run the optimization 24/7 enabling us to make every minute count.

Mr. Kiura: Now we are able to explore results in areas we couldn’t search before and find the design closer to the optimized result.

Could you provide some real case studies that show your use of modeFRONTIER?

Mr. Naase: I have been involved in the research of “EXlink”, the extended expansion linkage engine, as a mechanism for an extended expansion stroke in the multiple linkage system increases as compare to that of the conventional system (Fig. 1).

Mr. Kiura: We have been working on research and development toward enhancing the optimization of its efficiency in the internal combustion engine. This issue considers the phenomena and establishes an appropriate strategy. modeFRONTIER gives us a large amount of data and we continue our studies based on that data. For example, the number of principal design parameters was traditionally about 4 for the engine mechanism. In contrast, the number of principal design parameters for EXlink increased to 11 and we were at a loss to know that the parameter studies would be around 400 million designs. We ran about 1,000 to 2,000 calculations with a modelFRONTIER genetic algorithm and found the way to maximize performance. modelFRONTIER enabled us to run the optimization 24/7 enabling us to make every minute count.

Could you provide some real case studies that show your use of modeFRONTIER?

Mr. Naase: I have been involved in the research of “EXlink”, the extended expansion linkage engine, as a mechanism for an extended expansion stroke in the multiple linkage system increases as compare to that of the conventional system (Fig. 1).

Mr. Kiura: We have been working on research and development toward further improvement of thermal efficiency and reduction of emissions, noise, and vibration, of general purpose engines. We have established an engine performance diagnosis system by combining the optimization technology and response surface methods, engine combustion diagnosis technology, and numerical computation technology. This system offers an overview measurement of the engine performance by using the response surface method, identification of the control factors using optimization technology, measurement of flame, and clarification of the combustion mechanism by numerical computation.

What are your plans for using modeFRONTIER in the future?

Mr. Okubo: The problem that we have is increasing resistance and reducing the weight of engine components. Our intention is to achieve this through optimization. As there are more opportunities to use simulation technology, I would like to extend it to more users in our company.

Company Profile TAMA

TAMA is the acronym of Tecnologie Avanzate Miglioramento Ambientale (Advanced Technologies Environmental Improvement) and the company has been manufacturing filtration units, components and systems for air purification plants since 1990. TAMA products find important applications in many sectors: mineral, mechanical, wood, recycling, food, chemical, pharmaceutical, biomass and many others. TAMA main sold products are: filtration units for sheet metal working plants, baghouse filter unit Polaco-Air, cartridge filter unit Kompac-Air, Nooil FV filter, Wet filter Idromix, Active carbon adsorber, Cyclon separator, Silos, downdraft cutting tables for Laser and Plasma, Extractors, star valves, Archimedean screws, Soundproof box, Dry painting booth, Piping and various components.

In order to improve the current design process and to offer its customers products with excellent performance, TAMA has undertaken a new project including the use of structural and fluid-dynamics analyses, to enhance its products, taking advantage of the ANSYS simulation software. This system allows an in-depth analysis of both the airflow behaviour and of the structural behaviour of the filter itself.

Interview with Eng. Erik Chini, TAMA Technical Manager

The simulation as competitive advantage for a company producing air filtration units

Company Profile TAMA

TAMA is the acronym of Tecnologie Avanzate Miglioramento Ambientale (Advanced Technologies Environmental Improvement) and the company has been manufacturing filtration units, components and systems for air purification plants since 1990. TAMA products find important applications in many sectors: mineral, mechanical, wood, recycling, food, chemical, pharmaceutical, biomass and many others. TAMA main sold products are: filtration units for sheet metal working plants, baghouse filter unit Polaco-Air, cartridge filter unit Kompac-Air, Nooil FV filter, Wet filter Idromix, Active carbon adsorber, Cyclon separator, Silos, downdraft cutting tables for Laser and Plasma, Extractors, star valves, Archimedean screws, Soundproof box, Dry painting booth, Piping and various components.

1. Since when have you been using simulation technologies in your company?

Since 2012 we have gradually applied simulation to design, in relation to the several requirements coming from our customers or from our design team working on the product improvement. The objective in the short time is that of integrating the parametric simulation in our design process, since we have realized the importance of such technologies.

2. What was the main reason for introducing simulation three years ago?

Three years ago we started a new company project, strongly encouraged by our management, aiming at improving our products by means of simulation. We realized that our plants were often
oversized and in some cases we didn’t carried out an objective analysis, so to justify specific design decisions.

3. What kind of products are you using simulation for?
We carry out structural simulations on different plant components (panelling, brackets, plates...), mainly looking for the correct sizing of the plant itself and the optimization the material we use. We also do fluid-dynamic investigations, specially on big filters, allowing us to understand phenomena difficult to study with an experimental approach.

4. Does simulation effect your design process and how?
Simulation has proved to be a fundamental step for the production on high quality products, optimized and correctly sized according to customer’s needs and requirements.

5. Are you thinking about applying simulation also for new products production and what expectations do you have?
The design of our future products will be based on simulation technologies for sure. This will enable us to get the correct parameters for our plant sizing, thus guaranteeing the best productivity.

6. What's the contribution that EnginSoft can give you?
We know the technical value of a partner like EnginSoft that, thanks to its great multi-disciplinary experience, can support us specially when we have to deal with new challenges, facing not so known criticalities or in the case we could try to reach new markets.

7. Could you estimate the return on the investment related to simulation in your production?
Since we do not produce serial products, but on the contrary we customize every single product in accordance with the customer’s requirements, it is very difficult to estimate the quantitative return on the investment. In any case the correct plant and material sizing, that we get only by means of simulation, are the most important parameters for our company, in order to meet our customers’ needs and satisfaction.

An application case

Introduction
One of the first products investigated by means of the ANSYS software has been the filtration units for thermal cutting, as laser, flame cutting or plasma.

Starting from the standard geometry, consolidated in the company and designed with Autodesk Inventor, we have decided to carry out a series of structural and fluid-dynamic simulations, in order to highlight the criticalities and to identify the areas where an improvement of the product had to be pursued.

The validation of the results achieved with the simulation has been performed with some ad hoc measurements.

Fluid-dynamic Analysis

BOUNDARY CONDITIONS
In order to evaluate the flows inside the filtration chamber, the pre-chamber (labyrinth) and to understand where the powder normally accumulates, a CFD analysis has been performed using the following boundary conditions:

- Inlet flow rate equal to 4400 m³/h;
- Exit opening;
- Filtering elements considered as porous sub-elements;
- Momentum source corresponding to the fan as a function to its characteristic curve;
- Shear Stress Transport (SST) turbulence model.

RESULTS
A strong asymmetry of the flow can be observed. It occurs in the first chamber of the labyrinth, whose right part is affected only by circular flows and not by the main one; such behaviour has been observed also by experimentally performing some measurements in the section located in the centre of the inspection panel in the first room of the labyrinth.

Conclusions
Thanks to simulation, it has been possible to identify the areas where an intervention is necessary to improve the product performance.

In particular, as far as the structural analysis is concerned, we have realized that only some panels are really stressed, while others are totally unstressed and therefore thicknesses can be reduced and just few reinforcement plates should be placed where strictly necessary. The immediate consequence is a saving in material and money.

From a fluid-dynamic point of view, the achieved results have been particularly interesting since they proved that the chamber of pre-abatement is not so efficient, some internal obstacles increase the pressure drop and the sizing of the filter inlet section is not the correct one.

Thanks to these remarks we could modify and improve the product geometry.

All these evaluations have been possible thanks to the simulation tools that allows the reduction of design and prototyping time, thus providing an in-depth knowledge of some phenomena that cannot so easily be observed.

Ing. Erik Chini, TAMA Spa
On the frontier of drilling technologies

Tomax AS, a Norwegian supplier of drilling technology, develops tools to improve the drilling operations in challenging rock environments. Following the success of the Anti Stick-slip Tool, Tomax has developed and now qualifies a new drilling technology called the Afterburner. In order to optimize and improve the understanding of its performance, Tomax collaborated with EnginSoft and Flow Design Bureau (FDB) to investigate its fluid-dynamic behavior in laboratory and real world conditions through CFD modeling.

The Afterburner uses high-pressure mud to drive a number of ejector pumps to create a circulation loop between the well-bore annulus and the drillbit. This way, the solution removes problematic borehole erosion by the traditional high pressure jets and also creates a traction or pull force to the drillbit. The former effect allows a more gentle drilling operation even in unconsolidated rock formations, while the latter effect allows (at least in principle), wells of infinite length to be drilled. The technology enables the drilling operator (as opposed to conventional technologies), to establish stable wells in oil rich layers consisting of unconsolidated or loose rock. This allows a greater level of exploitation in oil fields with such layers, and ultimately increases the profitability the oil field’s reserves. The picture shows the drillbit assembly as tested at the Ulling test facility in Stavanger, Norway.

Although hugely positive results were obtained from a proof-of-concept test at a down-hole testing facility, the Afterburner required CFD analysis to establish its performance for an extended range of operating conditions. Typically, these conditions included variations in mud-properties and the complete bottom-hole-assembly. EnginSoft were tasked with validating the CFD model with the theoretical framework already developed by Tomax, and by tuning a few constants through CFD, a rather promising theoretical model capturing the overall performance emerged. The CFD model included Non-Newtonian fluid viscosities. In a detailed investigation of equipment, such as the Afterburner, the accurate prediction of fluid properties helped to determine the efficiency of rock-cuttings transport. The graph below compares the predicted CFD, analytically deduced and experimentally measured pressure differences over the Afterburner body. For constant viscosity fluids, an excellent match exists; while for mud fluids a deviation appears. The deviation occurs because of the inability of the analytical models to accurately represent the non-Newtonian fluid effects.

The Afterburner makes drilling possible in rock layers considered difficult to manage using conventional technologies, and therefore releasing reserves and increasing the value of the oil field. During the design of the Afterburner, Tomax demonstrated the concept and developed a theoretical foundation for many of the flow effects present. 3D CFD delivered by EnginSoft, allowed performance evaluation for conditions similar to those expected in actual applications. Furthermore, the collaboration between Tomax and EnginSoft/FDB added understanding and built a more solid theoretical framework for design evaluations and future improvements.

Nils Reimers, Director of R&D, Tomax AS.

“The patented Afterburner invention is based on re-routing the drilling fluids in a way never done before. The new routing takes care of several vital elements in a way that is not only new but also puts less on potentially unstable underground rock. This makes accurate modeling of the fluid dynamics the most critical element in the design process. The quality of the support provided by FDB and EnginSoft in this process has been paramount. FDB and EnginSoft responded very well to the challenge with great attention to detail and concept understanding. The result was an output that overlaid live recordings from the well. Consequently a great platform was established for optimizing the Afterburner invention for variations in fluid types and characteristics as well as borehole size and geometrical elements”

FDB also developed an analytical model for the leakage flow over the Afterburner. This model provided a framework to predict the effect of changing selected parameters. Combining the leakage model with the theoretical framework already developed by Tomax, and by tuning a few constants through CFD, a rather promising CAE EVENTS CALENDAR

Visit the EnginSoft event calendar and continue to advance your knowledge and skill set with innovative CAE Applications and Virtual Prototyping Technologies that can transform your design and development process with dedicated webinars and workshops around Europe.

Visit the EnginSoft event calendar and continue to advance your knowledge and skill set with innovative CAE Applications and Virtual Prototyping Technologies that can transform your design and development process with dedicated webinars and workshops around Europe.

Visit the EnginSoft event calendar and continue to advance your knowledge and skill set with innovative CAE Applications and Virtual Prototyping Technologies that can transform your design and development process with dedicated webinars and workshops around Europe.

Visit the EnginSoft event calendar and continue to advance your knowledge and skill set with innovative CAE Applications and Virtual Prototyping Technologies that can transform your design and development process with dedicated webinars and workshops around Europe.

Visit the EnginSoft event calendar and continue to advance your knowledge and skill set with innovative CAE Applications and Virtual Prototyping Technologies that can transform your design and development process with dedicated webinars and workshops around Europe.
Tolerance analysis on Emax 2 low voltage air circuit-breakers series

ABB SACE company profile

ABB (www.abb.com) is a leader in power and automation technologies that enable utility and industry customers to improve performance while lowering environmental impact. The ABB Group of companies operates in around 100 countries and employs about 150,000 people.

Product description

The power needed, when needed

SACE Emax 2 is a new series of low voltage air circuit-breakers available up to 6300 A and with the ability to efficiently and simply control electrical installations – from the traditional to the more complex – with minimum impact, the new SACE Emax 2 circuit-breakers represent the evolution of a circuit-breaker into a Power Manager.

SACE Emax 2 have been designed to increase efficiency in all installations: from industrial and naval applications to traditional and renewable power generation installations, buildings, data centers and shopping centers. Reliable protection and systems managed with competence.

Power Controller

The exclusive Power Controller function available on the new SACE Emax 2 circuit-breakers monitors the power managed by the circuit-breaker, keeping it below the limit set by the user. As a result of this more effective use, the peak of power consumed can be limited allowing savings on electricity bills.

The Power Controller, patented by ABB, disconnects non-priority utilities, such as for example electric car charging stations, during the times when consumption limits need to be respected, and connects them again as soon as it is appropriate. When required, it automatically activates auxiliary power supplies such as generator sets.

Ekip Touch

SACE Emax 2 circuit-breakers are equipped with a new generation of protection trip units that are easy to programme and read. The Ekip Touch trip units measure power and energy with precision and save the most recent alarms, events and measurements in order to prevent faults to the installation or trip effectively when necessary.

Ekip Link

The integrated IEC61850 communication module enables connection to automation systems and intelligent networks (Smart Grids). Accurate measurements of current, voltage, power and energy are all available by means of the communication modules and allow the trip units to be used as multimeters.

Network Analyzer

Upon request, the Network Analyzer function is also available, which controls the quality of absorbed power in real time and with extreme precision.

In addition, the innovative Ekip Touch and Hi Touch trip units in the G version include all the functions of generator protection switchgear, offering a safe control solution that is ready to use. No external devices, wiring and inspections are required.

Integration is easy. Even from afar

SACE Emax 2 series circuit-breakers have been designed to be integrated directly into all types of switchgear and automation and energy management systems to improve productivity and energy consumption. Complete integration into smart grids, in buildings and industrial plants is possible.

All circuit-breakers can be equipped with communication units for use with Modbus, Profibus and Devisenet protocols and with the modern Modbus TCP, Profinet and Ethernet IP protocols, which can be installed directly on the terminal box at any time.

The integrated IEC61850 communication module enables connection to automation systems and intelligent networks (Smart Grids). Accurate measurements of current, voltage, power and energy are all available by means of the communication modules and allow the trip units to be used as multimeters.

All circuit-breaker functions are also accessible via the Internet, in complete safety, through the Ekip Link switchgear supervision system and the Ekip Control Panel operator panel. The power and auxiliary connections are optimized to simplify connection to the switchgear.

The power terminals, which can be oriented horizontally or vertically, have been designed for the most common busbars, while the push-in connections of the auxiliaries ensure immediate and safe wiring.

Ekip Touch Emax 2 for the highest efficiency in panelbuilding

The customer

I.M.E.S.A. (based in Jesi, near Ancora - Italy) is a company that has been operating since 1972 in the field of electromechanical constructions.

I.M.E.S.A. is one of the leaders in Europe in the production of Low Voltage and Medium Voltage switchboards, SF6- insulated switch-disconnectors for internal and external installations for MV, supervision and control systems as well as turnkey electrical systems.

The challenge

In the context of a diesel-electric state-of-the-art ferryboat, I.M.E.S.A. faced the challenge of creating a switchgear column – feeding main distribution and motors- able to:

- Be fully selective with downstream circuit-breakers (Tmax T4)
- Grant a service short-circuit breaking capacity of 50 kA @ 600V AC
- Have a horizontal bussing distribution system
- Realize the most compact solution compatible with the use of withdrawable circuit-breakers.

These requirements originate by the need for extremely compact switchgear, a typical requirement that can arise from a marine environment.

The ABB solution

These requirements are really challenging. The first and second requirements clearly call for a category B circuit-breaker, while the third and the fourth are apter for a moulded-case circuit-breaker.
Model development and targets

Aim of the analysis performed by EnginSoft was the validation of the Emax 2 project. Validation means checking that product requirements are within their area of acceptability. Such a need is due to the fact that the propagation of the tolerances in assembled products doesn’t guarantee compliance with these requirements even when all parts composing the assembly were produced respecting the tolerances assigned to them by the designer.

Among others, three requirements of particular relevance have been identified for this product:

- Requirement on the capability of the single parts to be assembled together (assemblability).
- Requirement on the product quality.
- Requirement on security.

The procedure requires translating the above requirements in geometric entities in the CAD model, defining a range or threshold of acceptability for them and the assessment of their variability defined by the design (i.e. tolerances associated with nominal values).

The variability of functional measures is quantified after the analysis. The statistical analysis provides quantitative information on which take the most appropriate corrective actions and with minor impact on costs to improve the robustness of design. The entire procedure of construction of the model, its resolution, results visualization, their interpretation and appropriate corrective actions are handled by Cetol 6x, a dedicated software developed by the Sigmetrix. Cetol 6x automates the whole procedure following a standardized logical path remaining within the design environment, since it is fully embedded in the most popular CAD softwares.

Several practical advantages can be achieved: reduction of redesign cycles, prediction and reduction of the number of rejects, reduced reworks, fewer prototypes/samplings, dimensional control focused on the measures with major influence on product requirements, reduced “time to market”.

Obtained results and smart corrective actions

Assemblability requirement

The aim of the analysis is the provision of potential issues during the installation of the movable part with respect to the fixed one. The assessment of this requirement is based on the overall dimensions of the two mating parts and their difference.

The result obtained using the design specifications highlights a low probability (Fig. 2, approximately 3 products every 100) to get a negative gap between the supports and the guides of the movable part: this results in possible interferences and thus non-compliances during assembly phases.

The sensitivity analysis is a powerful tool that allows the identification of the dimensions and tolerances that, within the dimensional chain, have the greatest influence on the mean value and the width (standard deviation) of the resulting statistical distribution. The variables with the highest influence on the functional measure were found to be the tolerances associated with the thickness of the guide plates. They directly affect the width of the statistical distribution representing the gap in the mating.

The reduction of only two tolerances among the thousand in the model has allowed the vanishing of the non-compliances.

Product quality requirement

The aim of the analysis is the quantitative assessment of the gap between the plastic boxes held together and the frame of the movable part. During the life of a circuit breaker stressful condition can be encountered, i.e. high short circuit current passing through the breaker, when this happen the position of the phasors relative to each other and to the main structure is crucial to the performance. We must ensure a perfect fit in order to maximize results and reduce risks.

The result obtained using the design specifications highlights a high width for the output statistical distribution with respect to the ideal condition of operation, i.e. a mean value of zero for the gap and a null width of the distribution around the mean value.

Now, the targets to achieve are: shift the mean value of the distribution towards lower values and reduce its width. As previously happened, tolerance analysis has allowed the identification of the
variables with the greatest influence on the final requirement: results from sensitivity analysis have provided all the necessary information to act in a targeted manner. The tolerances belonging to the width of each box have the major influence on the dispersion of the gap value around the nominal, as a consequence it is on these that it was decided to take action.

The study has increased the robustness of the design through the reduction of only two tolerances (among thousands in the model) along with the change of a nominal value.

Security requirement

The aim of the analysis is the detection of possible risks of locking of the shutters during the extraction of the movable part. The closure of the shutters must always be ensured to avoid contact between the maintenance engineers and the parts under voltage of the switch.

The results obtained using the design specifications shows a potential number of non-compliances for the shutters in contact with the sliding guides causing friction and the subsequent locking.

The sensitivity analysis has identified the tolerances associated with the placement of two holes as the variables of greatest influence on the dispersion of the gap around the mean value.

The previous information could be used to undertake the appropriate corrective actions when products and parts are only on the drawings, avoiding their discovery during the prototyping phase and the costs associated with the production, quality controls and reworks. The reduction of only two tolerances among the thousands in the model allowed the avoidance of the non-compliances for the requirement under investigation.

Conclusions

The activity performed in collaboration with EnginSoft has allowed ABB to meet all the design requirements for the product under investigation. Through the analysis of the propagation of the tolerances within the process chain it has been possible to identify the dimensions and tolerances with the major impact on the functional measurements and therefore to act in a targeted way for the resolution of potential non-compliances, avoiding the generalization of their treatment. The resulting benefits are multiple, meeting objectives: reduction of the redesign cycles, savings on scraps and reworks, reduced number of prototypes, reduced “time to market”.

Giulio Tribulato - ABB SACE
Enrico Boesso - EnginSoft

Implementation of surrogate models to predict the morphology of membrane formation during solvent evaporation

Figure 6 - Initial statistical distribution of the distance between the shutters and the external wall

Figure 7 - Quantitative impact of variables on the result and its final statistical distribution

CETOL 6σ and Tolerance Analysis

EnginSoft is European distributor of leading Precise Constraint Technology software CETOL 6σ, developed by Sigmetrix LLC. CETOL 6σ is a tolerance analysis and optimization software tool providing the invaluable insight needed to address the above issues and to confidently release designs for manufacture. You will be able to assess the manufacturability of your designs long before they reach production.

CETOL 6σ enables designers and engineers to address multidimensional problems, by using a mathematical description of your CAD geometry. This unique method allows you to receive immediate analytical feedback, utilising the easy-to-use modeling, analysis and reporting components. The user is guided through the tolerance analysis and optimization process whilst being informed about missing or erroneous data.

Unlike simple 1D stack-up analysis or Monte Carlo Simulation, CETOL 6σ pursues a statistical approach employing advanced precision constraint technology, displayed in an intuitive graphical user interface. This approach accelerates the ability to identify part sensitivities and components that have the most significant contribute to the total variation. Discovering critical-to-quality assembly characteristics has never been easier.

For more information, visit http://www.tolerance-analysis.co.uk/products/cetol-60

Figure 1 - List of RSM algorithms available in modeFRONTIER 2014

Screening Analysis: SS-ANOVA

Even though this step is optional, it is very useful for gaining a better understanding of the model. The purpose of the screening is to detect the most important input variables, which is achieved by means of the Smoothing Spline ANOVA (SS-ANOVA) algorithm.

Model Validation

Cross-validation is an extremely popular methodology for verifying the prediction capabilities of a model generated from a set of samples. In cross-validation the data set is divided into L subsets, and each of these subsets is sequentially used as a testing set for a surrogate constructed on the other L–1 subsets. The prediction error can be estimated with all the L error measures obtained in this process (for example, as an average value).

METHODOLOGY

Import Designs

The Data Wizard of modeFRONTIER is used as the tool for importing data saved in Excel workbooks. After the input and output variables are selected, the following workflow is generated as in Figure 2 and is necessary to train the surrogate models.

The database can then be directly displayed within modeFRONTIER as in Figure 3.
2. Stepwise Regression

1. Smoothing Spline ANOVA

For each training table, the following surrogate models are trained: tables are created randomly to train the different surrogate models.

The training set is generated from the input table imported. Five training Surrogate Models displayed. In Figure 4 the ranking of the input variables for the Feret Ratio only is input variables (i.e. 100 designs) have been used to create a Testing set. A K-Fold method is favored with K=5 so that the 10 tables were created (5 Training and 5 Validation set).

RESULTS

Screening Analysis

The screening analysis was performed for each output data using the SS-ANOVA tool as described previously. In order to filter the most important variables, the cumulative effect was set to 90% for both the first and second order effects. The analysis is done with the input table and the results are displayed in Table 1 below.

Partition Tables

The input table is split into two or more partitions (i.e. row-wise). A set of designs for Training and Testing were specified to train the surrogate model and test its validity respectively.

80% of the designs of the input table were chosen randomly (i.e. 400 designs) as the Training set.

The other 20% designs (i.e. 100 designs) have been used to create a Testing set. A K-Fold method is favored with K=5 so that the 10 tables were created (5 Training and 5 Validation set).

Validation

The validation is done via the 5-FOLD approach. After training the whole set of surrogate models, the validation is done with the Testing data. The information is accessible directly from modeFRONTIER via the Validation Table tool. This tool is useful to visualize different kind of errors and to rank the surrogate models with respect to a chosen criterion for the error. In this study, the R-squared was selected as the criterion and the average R-squared of the 5 validation tables is used to select the best surrogate model, i.e. the one for which the R-squared is closest to 1.

Export Surrogate Models

This section deals with exporting the surrogate model to an Excel spreadsheet and explains how to use it properly.

As there are multiples outputs, it has been decided to use Default parameters in order to quickly assess which surrogate models are the most efficient.

As a consequence, for each output (8) and for each training table (5), 19 surrogate models were trained that is to say a total of 760 surrogate models.

The most important input parameters selected to train the surrogate model were parameterized as follow:

- Smoothing Spline ANOVA - Internal Optimization activated for the main and interaction effects modes
- Stepwise Regression - The degree can vary from 1 to 5.

Most of these surrogate models are trained with their parameters set to Default. Only the Smoothing Spline ANOVA and the Stepwise Regression were parameterized as follow:

1. Smoothing Spline ANOVA - Internal Optimization activated for the main and interaction effects modes
2. Stepwise Regression - The degree can vary from 1 to 5.

As there are multiples outputs, it has been decided to use Default parameters in order to quickly assess which surrogate models are the most efficient.

As a consequence, for each output (8) and for each training table (5), 19 surrogate models were trained that is to say a total of 760 surrogate models.

In this study, the surrogate model was exported to a macro enabled Excel Workbook. The Excel file contains 3 sheets as displayed below.

Conclusions

The most important input parameters selected to train the surrogate model upon in modeFRONTIER 2014. When the surrogate model is not sufficiently accurate, one can either add or remove some input variables and check the sensitivity of the mean R-squared. The mean R-squared was computed via five different validation table in order to check the robustness of each of the surrogate models. The surrogate model with the best performance was selected and export to an Excel file in order to be used independently of modeFRONTIER.

The input variables that were used for each surrogate models are displayed below.

Finally, the surrogate models that were chosen for each output are displayed below.

It has to be noticed that the Feret Ratio surrogate model does not perform well, so that the user should rather use the Max and Min Feret in order to deduce the Feret Ratio.

These surrogate models can be used to identify trends or to perform virtual optimization. Instead, these surrogate models are not necessarily accurate in predicting the value of an output and depends upon the quality of the training points and choice of surrogate model.

However, if the user wants to predict an output very quickly, one can use these surrogate models, but it should be kept in mind that the output value has to be verified with data coming from either a simulation and/or experimental test.

Romain Soulier - EnginSoft France

M. Pluton Pulumbi - Air Liquide

Case Histories

Case Histories

Figure 2 - Workflow

Figure 3 - Extract of the design table

Table 1 - Selected input variables for each output (in red the non selected input variables)

Table 2 - Selected input variables for each output

Table 3 - Selected surrogate models for each output

Figure 4 - Main & Interaction Effect - Feret Ratio

Figure 5 - Export Surrogate Model - Excel File - 1st sheet

Figure 6 - Export Surrogate Model - Excel File - 2nd sheet

Figure 7 - Export Surrogate Model - Excel File - 3rd sheet

Figure 8 - Export Surrogate Model - Excel File - 4th sheet

Figure 9 - Export Surrogate Model - Excel File - 5th sheet

Figure 10 - Export Surrogate Model - Excel File - 6th sheet

Figure 11 - Export Surrogate Model - Excel File - 7th sheet

Figure 12 - Export Surrogate Model - Excel File - 8th sheet

Figure 13 - Export Surrogate Model - Excel File - 9th sheet

Figure 14 - Export Surrogate Model - Excel File - 10th sheet

Figure 15 - Export Surrogate Model - Excel File - 11th sheet

Figure 16 - Export Surrogate Model - Excel File - 12th sheet

Figure 17 - Export Surrogate Model - Excel File - 13th sheet

Figure 18 - Export Surrogate Model - Excel File - 14th sheet

Figure 19 - Export Surrogate Model - Excel File - 15th sheet

Figure 20 - Export Surrogate Model - Excel File - 16th sheet

Figure 21 - Export Surrogate Model - Excel File - 17th sheet

Figure 22 - Export Surrogate Model - Excel File - 18th sheet

Figure 23 - Export Surrogate Model - Excel File - 19th sheet

Figure 24 - Export Surrogate Model - Excel File - 20th sheet

Figure 25 - Export Surrogate Model - Excel File - 21st sheet

Figure 26 - Export Surrogate Model - Excel File - 22nd sheet

Figure 27 - Export Surrogate Model - Excel File - 23rd sheet

Figure 28 - Export Surrogate Model - Excel File - 24th sheet

Figure 29 - Export Surrogate Model - Excel File - 25th sheet

Figure 30 - Export Surrogate Model - Excel File - 26th sheet

Figure 31 - Export Surrogate Model - Excel File - 27th sheet

Figure 32 - Export Surrogate Model - Excel File - 28th sheet

Figure 33 - Export Surrogate Model - Excel File - 29th sheet

Figure 34 - Export Surrogate Model - Excel File - 30th sheet

Figure 35 - Export Surrogate Model - Excel File - 31st sheet

Figure 36 - Export Surrogate Model - Excel File - 32nd sheet

Figure 37 - Export Surrogate Model - Excel File - 33rd sheet

Figure 38 - Export Surrogate Model - Excel File - 34th sheet

Figure 39 - Export Surrogate Model - Excel File - 35th sheet

Figure 40 - Export Surrogate Model - Excel File - 36th sheet

Figure 41 - Export Surrogate Model - Excel File - 37th sheet

Figure 42 - Export Surrogate Model - Excel File - 38th sheet

Figure 43 - Export Surrogate Model - Excel File - 39th sheet

Figure 44 - Export Surrogate Model - Excel File - 40th sheet

Figure 45 - Export Surrogate Model - Excel File - 41st sheet

Figure 46 - Export Surrogate Model - Excel File - 42nd sheet

Figure 47 - Export Surrogate Model - Excel File - 43rd sheet

Figure 48 - Export Surrogate Model - Excel File - 44th sheet

Figure 49 - Export Surrogate Model - Excel File - 45th sheet

Figure 50 - Export Surrogate Model - Excel File - 46th sheet

Figure 51 - Export Surrogate Model - Excel File - 47th sheet

Figure 52 - Export Surrogate Model - Excel File - 48th sheet

Figure 53 - Export Surrogate Model - Excel File - 49th sheet

Figure 54 - Export Surrogate Model - Excel File - 50th sheet

Figure 55 - Export Surrogate Model - Excel File - 51st sheet

Figure 56 - Export Surrogate Model - Excel File - 52nd sheet

Figure 57 - Export Surrogate Model - Excel File - 53rd sheet

Figure 58 - Export Surrogate Model - Excel File - 54th sheet

Figure 59 - Export Surrogate Model - Excel File - 55th sheet

Figure 60 - Export Surrogate Model - Excel File - 56th sheet

Figure 61 - Export Surrogate Model - Excel File - 57th sheet

Figure 62 - Export Surrogate Model - Excel File - 58th sheet

Figure 63 - Export Surrogate Model - Excel File - 59th sheet

Figure 64 - Export Surrogate Model - Excel File - 60th sheet

Figure 65 - Export Surrogate Model - Excel File - 61st sheet

Figure 66 - Export Surrogate Model - Excel File - 62nd sheet

Figure 67 - Export Surrogate Model - Excel File - 63rd sheet

Figure 68 - Export Surrogate Model - Excel File - 64th sheet

Figure 69 - Export Surrogate Model - Excel File - 65th sheet

Figure 70 - Export Surrogate Model - Excel File - 66th sheet

Figure 71 - Export Surrogate Model - Excel File - 67th sheet

Figure 72 - Export Surrogate Model - Excel File - 68th sheet

Figure 73 - Export Surrogate Model - Excel File - 69th sheet

Figure 74 - Export Surrogate Model - Excel File - 70th sheet

Figure 75 - Export Surrogate Model - Excel File - 71st sheet

Figure 76 - Export Surrogate Model - Excel File - 72nd sheet

Figure 77 - Export Surrogate Model - Excel File - 73rd sheet

Figure 78 - Export Surrogate Model - Excel File - 74th sheet

Figure 79 - Export Surrogate Model - Excel File - 75th sheet

Figure 80 - Export Surrogate Model - Excel File - 76th sheet
Multi-Objective Optimization of XBA Sentinel Antenna

A multi-objective optimization approach has been applied to the synthesis problem of the geometrical structure of antenna with complex set of requirement specification. This methodology is a promising tool for new development and application for several kinds of microwave structure.

A typical optimization problem regarding spacecraft antennas has been considered to improve the communication capabilities of an antenna used for down link data-handling (PDHT). These antennas play an important role in many mission of Earth Observation from Space, where high transmission rate is necessary to acquire Earth images in various spectral bands for an enormous number of civilian and military applications. Thales Alenia Space first exemplary of a PDHT antenna was developed for RADARSAT and COSMO-SKY Med missions about ten years ago. For recent satellite missions new and more stringent performances are requested for the antenna pattern, especially in terms of cross polarization discrimination and high transmission rate is necessary to acquire Earth images in various spectral bands for an enormous number of civilian and military applications. Thales Alenia Space first exemplary of a PDHT antenna was developed for RADARSAT and COSMO-SKY Med missions about ten years ago. For recent satellite missions new and more stringent performances are requested for the antenna pattern, especially in terms of cross polarization discrimination, amplitude and phase ripples in band, return loss (Figure 1). The basic antenna architecture is a symmetric cylindrical corrugated surface fed by quartz loaded open waveguide launcher (Figure 2), so that the analysis is based on simple 2D Method of Moment modelling, while the optimization is performed by a quasi-Newton minimization techniques. More stringent antenna requirements entail a redesign of the electromagnetic/geometrical structure: to improve the electrical performance a series of slots across the corrugation walls have been inserted (Figure 3) The break on the geometrical symmetry requires now more complex 3D e.m. description with large increasing on computation time and memory resources. The global geometry definition should require over 100 independent variables, so that the possibility to optimize the antenna performance becomes very demanding, taking into account 6 hours for a single computation run (4 CPUs Xenon at 3.0 GHz)

The multi-objective/multivariable nature of the optimization problem derives from multiple requirements on electrical antenna performances, with appropriate weight if necessary. These mutually dependent RF parameters derive from operative parameters as satellite and antenna position, visibility angle, pointing error angle, minimum gain at nadir, etc.

Gain requirement defined with respect a mask in a well defined angular range and in a particular frequency bandwidth;

- Gain ripple requirement with respect the frequency;
- Phase Ripple requirement (quadratic component of the gain) with respect the frequency;
- XPD requirement defined, in the same manner as the gain, in specified angular and frequency ranges;
- Return Loss requirement;
- Mechanical feasibility of the theoretical geometrical structure yields very difficult to explore the complex solution space in order to obtain the best geometrical structure. Because no analytical models are available for this complex structure the only possible way to solve the problem is the numerical approach. The most important topics in the optimization problems are:

The powerful capabilities of the modeFRONTIER optimization definition of the objective function;

- The identification of solutions that goes toward the convergence of the objective functions.
- “minimal approach” with a geometrical structure profiled by an established function (polynomial function, trigonometric function) in general described by few variables.
- “aimed approach” that divides the variables in homogeneous classes and acts only on one group per time, appropriately individuated.

Taking into account the particular nature of the electromagnetic problem the range for the variation of geometrical dimensions has been established into a half wavelength. The starting point for the optimization has been chosen taking into account the experience matured on the previous PDHT design activities configuration.
it post-processes the Microwave Studio CST results in order to compute the relevant error functions.

Optimization
The individuation of the more sensitive variables has been an important step in the optimization starting phase. A DOE (Design of Experiment) technique has been used to understand better the input-output relationship and to reduce the design space dimension (number of variables) combining Sobol and Latin Square methodologies. A correlation scheme among the optimization variables is reported in Figure 5.

The correlation map of the variables considered in the optimization gives information on the dependence among the parameters (Figure 8). For example the parameters para1, para2 and para3 have the effect to modify the surface profile, while extlen sets the position of the launcher with respect to surface. The high correlation coefficient of extlen with para2 and para3 gives the evidence of the similar physical effect that these variables have on the output variables considered.

In Figure 6 the history chart is reported for the main electrical parameters (copular mask and XPD). The result shows that id 152 is the better compromise for these two goals. The same result is reported in the bubble chart with more evidence of zone near the Pareto frontier (Figure 7).

Results
More performing solutions with respect to requirements have been individuated by the optimization process. Among the more challenging solutions the selected one has a high degree of regularity in terms of corrugation depth, corrugation steps, dimension of the coupling slot. These results have been reached by the powerful modeFRONTIER capability to individuate the sensibility between the geometrical data and performance (or in other words the different objective functions).

With regards to the desired performance modeFRONTIER has been able to obtain the following improvement with respect to old configuration, in particular:

- The gain mask has been fully satisfied, with desired improvement of the gain at 62° (6.6 dB now w.r.t. 6 db) (see Figure 8);
- The pattern widening on the enlarged coverage has been achieved so that the antenna can be used for the lower satellite position with 70 degs field of view;
- The XPD increase of 7 dB passing from 5 dB to 12 dB;
- The phase ripple in carrier band is satisfied with 3 deg (peak to peak) variation. The new solution presents a equalized copular pattern over a large frequency bandwidth;
- The amplitude ripple in carrier band has been satisfied with 1 dB (peak to peak) variation. The new solution presents a equalized copular pattern over a large frequency bandwidth;
- The phase ripple in carrier band is satisfied with 3 deg (peak to peak) variation.

It is evident a good improvement of XPD in the central coverage region comprise between ±50 degs. In the final antenna structure the cross-polar peak has been reduced to -15 dB in comparison to -5 dB with plus 10 dB decreasing. Another very interesting improvement on the antenna diameter is could be possible: it has experienced during the optimization process a reduction of the diameter is reachable. Antenna geometry with a 300 mm diameter could be able to maintain the same performance (with the only drawback to have a minor pattern slope out of the coverage).

The reduction of the size has been showed as a new solution with good advantages on the antenna satellite allocation.

Conclusions
The potential of the combined use of modeFRONTIER with electromagnetic tools (such as Microwave Studio by CST) has been demonstrated for a fuselux antenna synthesis, which constitutes a very complex e.m. problem, especially when low XPD is requested on a wide-coverage. This procedure could be a first step in the development of more advanced design procedures based on the modeFRONTIER environment. In particular it has to emphasize:

- Research of the theoretical performance limits for e.m. configuration (typical research of Pareto frontier);
- Individualization of the robust design in terms of sensibility analysis, envelope minimization and weigh (important for space application);
- Individualization of the transfer functions (using the modeFRONTIER capabilities of the Response Surface Methodology) as system response to permit the their use in other complex systems (smart antenna system, reconfigurable antenna, multibeam adaptive antenna systems);
- Material characterization and parameters extraction and technological process refinement.

Acknowledgement
The authors would like to thank Mr Peter Rinous, ESA for valuable suggestions on antenna development.

R. Ravarelli - Thales Alenia Italia Co.
C. Iannicelli - Software System Engineering Co.
F. Franchini - EnginSoft
Magnetostatic Analysis on ITER Test Blanket Modules

ITER (acronym of International Thermonuclear Experimental Reactor) is an international nuclear fusion research and engineering project, which is building the world’s largest experimental Tokamak nuclear fusion reactor in the south of France. The ITER project aims to make the long-awaited transition from experimental studies of plasma physics to full-scale electricity-producing fusion power plants. In Figure 1 a cross section of ITER is depicted. The Tokamak concept is based on the magnetic confinement, in which the plasma is contained in a doughnut-shaped vacuum vessel. The fuel, a mixture of deuterium and tritium, two isotopes of hydrogen, is heated to temperatures in excess of 150 million °C, forming a hot plasma. Inside the plasma the nuclear reaction takes place: due to the high kinetic energy the deuterium and tritium nuclei collide, fuse into heavier helium atoms and release tremendous amounts of energy in the process. In the Figure 2 a simplified reaction scheme is depicted.

Aim of the analysis

In the present document a magnetostatic analysis on new test blanket modules is illustrated. Europe is currently developing two Test Blanket Modules (TBMs): the Helium-Cooled Lithium-lead (HCLL) concept which uses the Lithium-lead as both breeder and neutron multiplier, and the Helium-Cooled Pebble-Bed (HCPB) concept which features lithium ceramic pebbles as breeder and beryllium pebbles as neutron multiplier. Both concepts use Reduced Activation Ferritic Martensitic (RAFM) steel as structural material, the EUROFER.

The TBMs of both HCLL and HCPB concepts will be inserted in an a system of pipes, components, and supporting structures located inside the Port Cell. In Figure 3 the test blanket modules are depicted. In brief the need to feed, during the reaction, the plasma with Tritium led ITER designers to test new blanket modules inside the vacuum vessel made by ferromagnetic materials. Materials that can be magnetized, which are also the ones that are strongly attracted to a magnet, are called ferromagnetic (or ferrimagnetic).

The presence of ferromagnetic material leads to the developing of magnetic forces on the blanket module surfaces that are exposed to the vacuum vessel due to the difference in magnetic permeability between the ferromagnetic material of blanket modules and air. Thus, it is crucial to assess these forces by means of a FEM calculation. The presence of ferromagnetic material leads to the developing of magnetic forces on the blanket module surfaces that are exposed to the vacuum vessel due to the difference in magnetic permeability between the ferromagnetic material of blanket modules and air. Thus, it is crucial to assess these forces by means of a FEM calculation. The Boundary condition. As described, aim of the simulation is the calculation of Forces and Moments on TBMs during operational condition. The calculation is made by ANSYS Maxwell and ANSYS EMAG to further comparison. In this document the Maxwell simulation is described along with a comparison with EMAG solution.

Geometric assumptions for FEM modelling.
The two TBMs were modeled as two solid blocks with dimensions as described in Figure 4. The distance between the two TBMs is 0.25 m. The coordinates of the vertex A and B are $x = 6.5m$ and $z = 0.2m$, in a cylindrical coordinate system with origin in the centre of the ITER machine. It is assumed that the TBMs are placed inside each one of the 18 equatorial ports (between two consecutive TF coils) in a symmetric position with respect to the port poloidal midplane.

Material properties.
The B-H curve data for the TBMs are supplied by EUROFER. It was assumed that in the HCPB the ratio metal/no metal is 0.8, while in the HCLL it is 0.6, therefore equivalent smeared properties were used. To get the smeared BH curve table data the following function has been used:

$$B_{smeared} = \mu_0 \left( H + \frac{\nu_{sat}}{\nu_{mod}} \right)$$

This function calculates, for each point of the original BH curve, the correspondent value of the smeared BH curve implementing the ratio between metal/no metal of the considered object.

Since at the present state of the design TBMs geometry is not completely defined the use of smeared materials property is acceptable for the TBMs feasibility study.

Loads and boundary conditions

To simulate the operational conditions all the superconducting coil system is modeled along with the plasma current. The coil system is made of three kinds of coils: Poloidal Field (PF), Toroidal Field (TF) and Central Solenoid (CS). In Figure 5 the superconducting coil system is depicted.

The Maxwell model

Maxwell is a software package for low-frequency electromagnetic field simulation. Maxwell can be used to design 3-D/2-D structures, such as motors, actuators, transformers and other electromagnetic and electromechanical devices. Maxwell uses the Ansoft-pioneered automatic adaptive meshing capability which automatically creates and refines the finite element mesh as the solution converges, streamlining the solution process and making the software easy to use.

The model

The geometry of the model is completely made inside Maxwell. Geometrical parameters are used to take into account, in case, both different dimensions of TBMs and their position compared to plasma current. Due to geometry and load symmetry 20° of the whole machine is considered. In Figure 6 the Maxwell model is depicted along with the applied boundary conditions.

FEM model

The functionality of the auto-adaptive mesh algorithm is tested. Two mesh models are built. The first one implements only the auto-adaptive meshing. This way 16 passes are computed and the number of 363996 elements is reached. The second model is carried out by the means of mesh settings in order to refine the mesh on blanket modules. In particular a body sizing of 0.08 m is imposed. This way just 3 passes are computed and 754766 elements are used to complete the FEM model.

Simulations show the same result between the two models adopted although, due to a lower number of elements, the first method is faster. This example shows the reliability and correctness of the auto-adaptive mesh algorithm that achieves the best compromise between the accuracy of the solution and the speed-up of the simulation. In Figure 7 a comparison between the values of radial force on HCLL blanket module for the two mesh methods adopted is depicted.

Results

In Figure 8 a comparison between the B-field calculated by Maxwell and EMAG (the legacy portion of ANSYS FEM code for low frequency electromagnetic analysis) on the whole models is depicted. In Figure 9 the B-field is shown on the blanket modules.
the construction of demonstration fusion reactors. 

F4E also supports fusion research and development in Japan, China, India, South Korea and the United States. 

Conclusion 

Considering the problem is not confined and the magnetic steel is in saturation, the two software show a good agreement with regard to both field plots and force values. The order of magnitude of the calculated forces provides fundamental information with regard to the IFMs feasibility study.

Fusion For Energy 

Fusion for Energy (F4E) is the European Union’s Joint Undertaking for ITER and the Development of Fusion Energy. The organisation was created under the Euratom Treaty by a decision of the Council of the European Union in order to meet three objectives: 

- F4E is responsible for providing Europe’s contribution to ITER, the world’s largest scientific partnership that aims to demonstrate fusion as a viable and sustainable source of energy. ITER brings together seven parties that represent half of the world’s population – the EU, Russia, Japan, China, India, South Korea and the United States. 

- F4E supports fusion research and development initiatives through the Broader Approach Agreement, signed with Japan – a fusion energy partnership which will last for 10 years. 

- Ultimately, F4E will contribute towards the construction of demonstration fusion reactors. 

Case Histories 

This work was presented at the recent TechNet Alliance meeting in Aix-en-Provence. The meeting was largely about analysis, simulation and experimental testing in the design of ITER components, and was opened by Cornelis Jong from the ITER organisation. Contributions were offered by companies throughout the world, including TaeSung S.&.E, Korea, CADFEM CIS, Russia, DAES SA, Switzerland, and many others.

Benelli’s Mission: technology, quality, performance and design 

Continuous research, innovation and development of new technologies, materials, precision engineering and distinctive designs are the keystones of Benelli’s philosophy. Benelli’s mission is to design and manufacture shotguns that stand out from the competition by their advanced technology, refined styling and unrivalled reliability. To improve the shotgun performance, Benelli has introduced modelFRONTIER into their product development cycle. 

The first project to adopt modelFRONTIER was the optimization of the Progressive Comfort to be used in the new 828U over and under shotgun. This was developed from the standard Progressive Comfort, already in production and sold in the semi-automatic market.

Introduction 

For hunters, a shotgun’s lightness and overall dimension are very important. A low weight ensures its easy portability and gives good handling during a day’s hunt, while keeping each component’s dimension small allows more design freedom for neighboring components, especially inside the stock where there is very limited space.

Starting from these considerations, the goal was to reduce the weight and the overall dimension of the standard Progressive Comfort already in production while maintaining the same mechanical behavior (the same force-deformation curve) for the new 828U over and under shotgun.

Structural optimization of the Benelli Progressive Comfort 

Dimensional and weight optimization under maximum cartridge load 

TechNet Alliance is perhaps the world’s largest network in the Engineering Simulation community, offering software technologies, competencies, training and education, support at all levels, as well as a panel of professionals from industry, academia, and technology providers. TechNet members meet on regular basis to exchange experiences, information and knowledge. For more info see: http://www.technet-alliance.com

Benelli S.r.l. 

The Progressive Comfort description 

The standard Progressive Comfort (figure 1) is a mechanical system designed to reduce the impact force on the operator’s shoulder during fire. It is composed of three stages of beams in parallel, where each stage has a couple of beams in order to guarantee a progressive stiffness as load increases (low, medium and high energy cartridges - Figure 2). 

Workflow description 

Starting from the standard Progressive Comfort, the optimization work was split into three parts, one for each stage of beams, since the three stages are structurally decoupled. A unique workflow was built with three different constraint values for the maximum stage load. The workflow linked to a 2D CAD model (built with...
The software UG NX 8.5) thanks to the symmetry of the system, just one half of the entire progressive comfort was modelled with a plain strain approach. For the computation of the mechanical characteristics MSC Patran was used as a pre and post processor and MSC Marc as the non-linear structural solver.

The FEA was a non-linear static analysis with imposed displacement to ensure the same mechanical behavior was obtained (the same force-deformation curve) as the standard Progressive Comfort. Due to the absence of a direct MSC Patran node, two routines (files, ses) were written and called by two DOS nodes, one to automate the pre-processor process (meshing, loads, constraints, files for MSC Marc data input, etc.), and another one to automate the post-processor process, in order to export all the mechanical characteristic needed to optimize the structure (stress, characteristic etc.).

The workflow nodes are:
- N.10 geometrical input variables (beam length, beam width, blends, …)
- N.11 output variables (total weight, overall dimension, stress, strength, …)
- N.2 design objective (mass, overall dimension)
- N.3 constrains (maximum width, maximum stage load, admissible stress)

The output variables were taken from the CAD model via a direct node (total weight, overall dimension, etc), outputs from the CAE model via a DOS node (stress, deformation etc.) and by the calculator node (Figure 3). The optimization strategy started from a DOE of 44 Sobol designs and used 22 subsequent generations created with the NSGA-II algorithm. This gave a total number of 968 design configurations evaluated for each stage. The simulation time for each run was about 1 minute (thanks to the simplification of bi-dimensional CAD model and of plain strain approach) therefore the total time for each stage was about 13 hours and for the entire structure (three stages) 39 hours.

Results
By plotting the two design objectives (mass, overall dimension) on the scatter chart and marking the Pareto designs we chose for each stage of beams the best solution giving a major weight to the overall dimension reduction than to the mass. (Figures 4-5-6).

By combining the results of the three stages a new optimized Progressive Comfort was obtained with the following characteristics:
- Overall dimension reduction of 35%
- Weight reduction of 36%
- Same mechanical behavior (stress, force, displacement) in relation to the standard ones.

By comparing the design given by modeFRONTIER to a manually derived design the following benefits were achieved:
- more configurations were evaluated (2895 vs 147),
- a lot of time was saved in the model management of CAD-CAE (20 man/ hours vs 100 man/hours)
- a lot of post-processing time was saved (3man/hours vs 50 man/hours).
- the time machine was increased (50 hours vs 3 hours).
- utilization of down time during night or week end was increased (70% vs 2%).
- time to market of the product was reduced (1 month vs 4 months).

The usage of modeFRONTIER allowed the hardware resources to be maximized, to build an efficient historical data store and to improve the Benelli know how on the Progressive Comfort System.
Coffee Makers and Homecare Products with a Combination of style and performance

De’ Longhi is a great Italian group, growing in the world thanks to its innovative products, its style and quality of materials. It is a global market leader operating in the comfort sector with four different business units: heating, air conditioning, air treatment, kitchen appliances, home care and ironing.

De’ Longhi is a successful brand thanks to its ideas and innovation and the company has been able to extend its products portfolio from portable heating (oil filled radiators, convector heaters, fan heaters, gas and kerosene heaters, etc.) to air conditioning, a sector in which De’ Longhi offers the widest range of possible products, from Pinguino till the chilling and air conditioning systems for industrial civil uses (shopping malls, hospitals, banks and hotels).

De’ Longhi products now include air treatment systems, homecare and civil uses (shopping malls, hospitals, banks and hotels), heat exchangers and air conditioning, a sector in which De’ Longhi offers the widest range of possible products, from Pinguino till the chilling and air conditioning systems for industrial civil uses (shopping malls, hospitals, banks and hotels).

The wide range of De’ Longhi products is the result of a mixture of productive, technological and commercial synergies and its evolution is supported by strong investments for Research & Development and Industrial Design, so to make an idea to become a product. Our strength mainly lays in the constant technological improvement of the single productive units and in the great care for the most critical phases of the industrial process, so to maximize productivity and quality, while differentiating the products and reducing their time-to-market. In such a scenario, the use of top class technologies like ANSYS and ANSYS CFX, also thanks to EnginSoft’s experience, has allowed us to move forward our objectives of innovation and quality even in the experimental phase.

Stefano Quaresimin, Raffaele Toffolo, Marco Frasson
Research & Development, De’ Longhi Spa

EnginSoft in the food&beverage industry

In addition to the 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, aerobic 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, and try to enhance the time-to-market. In such a scenario, the use of top class technologies like ANSYS and ANSYS CFX, also thanks to EnginSoft’s experience, has allowed us to move forward our objectives of innovation and quality even in the experimental phase.

Stefano Quaresimin, Raffaele Toffolo, Marco Frasson
Research & Development, De’ Longhi Spa

Why ANSYS and ANSYS CFX in De’ Longhi

We use the ANSYS NLS code to verify and optimize components for coffee makers and ironing systems. ANSYS is applied in the first phases of the product design or to enhance existing products, guaranteeing that the project complies with the necessary performance requirements before its production starts. We carry out structural, thermal and non-linear analyses, mainly on components made of polymeric material. ANSYS CFX is instead used to optimize the heat exchange in our heating appliances with the aim of reducing the service temperatures and in particular on those surfaces that could get in contact with the customer. ANSYS CFX allow us to reduce the number of prototypes which are necessary to guarantee the correct attainment to the regulation and to validate the project functionality at the same time.

Apart from the above-mentioned advantage, having a new tool of this kind in our working chain, enables us to reduce the time to market for a new product. Furthermore, the variety of scenarios that this tool can investigate, allows to limit the number of returned goods in warranty.

EnginSoft in the food&beverage industry

In addition to the 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, aerobic 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 be maximized. 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 expertise 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.
Optimization of the production process of a die-casting component for automotive using MAGMAfrontier

One of the trends that are characterizing the market of the automotive components concerns the growing application of the cold chamber die-casting as casting production process. The reason is related to its cheapness, due to a high productive frequency and to the precision of the producible components. On the other hand, the particularly high velocities, that rule the process, affect the filling dynamics, characterizing it with turbulence effects that can lead to defects generation and degenerate into air entrapments or not completely filled parts.

In such a context, the product/process design assumes an extremely important role, since it’s a very delicate phase in which the most effective solutions are evaluated in order to produce the equipment and to optimize the proper process parameters. Development and Optimization of a productive process means to identify the variables that mainly impact on the product features, evaluating the effects with the support of advanced tools for process simulation.

This work describes how Tecnopress S.p.A., leader in die-casting components supply on a worldwide level, has applied the numerical optimization so to enhance the control on the production process of an automotive component, allowing to reaichie expected qualitative level without affecting the production cost.

Introduction

The actual market conditions force companies to react in a very short time, producing castings with high qualitative characteristics and competitive prices. In order to meet all requests, component suppliers need productive processes with high productive frequency, requiring a minimum effort of machining for finishing (net-shape parts), to be obtained for example with a high operating reliability in accordance with the strict qualitative tests carried out in any production phase from casting to final assembly.

The consolidated approach to casting adopted by Tecnopress S.p.A. technical staff consists of a preliminary analysis of the component, so to define the most suitable configuration according to the features of the available company installations. Once the basic equipment configuration has been fixed (nr. of figures, parting lines, lateral slides, couplings and dimples location, etc.), the adequate casting system is defined, that according to the experience should provide a good cavity filling. Tecnopress S.p.A. has strongly integrated the casting simulation with MAGMA5 code in its standard design workflow for a long time now, so to rely on robust and reliable tools, able to guarantee its customers’ with high quality products.

In this specific case, having no qualitative requirements provided by the customer, and considering the very short time for the first sampling production, it has been decided to start with a simple test, with a virtual analysis of the filling of the current design status. The analysis of the results obtained by the first simulation has satisfied the design expectations on velocity and temperature distribution during filling but has highlighted some criticalities due to air entrapment in some casting areas (Figure 1). The indications provided by the virtual analysis has suggested that the dimples distribution during filling has highlighted some criticalities due to air entrapment in some casting areas (Figure 1). The indications provided by the virtual analysis has suggested that the dimples location and/or poor compaction of the solidification shrinkage.

Preliminary SAMPLING of the castings

The production of the new equipment and the process parameters of the first sampling reflect the setup of the virtual simulation. Although the castings produced during the first sampling do not present particular defects on a visual inspection at a rough state, the pressure tightness tests (up to 13Bar in the air) after machining have produced a 30% of production scraps.

Design of the die/equipment/casting system

The casting, as a result of the analyzed process, consists of a cover to obtain a pneumatic valve necessary to make a braking system operate. Heavy tracks (tracks, tractors, trailers, etc.) are commonly equipped with this kind of system. This component is not affected by heavy mechanical stress, but has to ensure a perfect tightness both on the working areas, where linings are often located, and on under pressure surfaces. It’s therefore a component of a safety system that has to guarantee a high operating reliability in accordance with the strict qualitative tests carried out in any production phase from casting to final assembly.

The consolidated approach to casting adopted by Tecnopress S.p.A. technical staff consists of a preliminary analysis of the component, so to define the most suitable configuration according to the features of the available company installations. Once the basic equipment configuration has been fixed (nr. of figures, parting lines, lateral slides, couplings and dimples location, etc.), the adequate casting system is defined, that according to the experience should provide a good cavity filling. Tecnopress S.p.A. has strongly integrated the casting simulation with MAGMA5 code in its standard design workflow for a long time now, so to rely on robust and reliable tools, able to guarantee its customers’ with high quality products.

In this specific case, having no qualitative requirements provided by the customer, and considering the very short time for the first sampling production, it has been decided to start with a simple test, with a virtual analysis of the filling of the current design status. The analysis of the results obtained by the first simulation has satisfied the design expectations on velocity and temperature distribution during filling but has highlighted some criticalities due to air entrapment in some casting areas (Figure 1). The indications provided by the virtual analysis has suggested that the dimples location and/or poor compaction of the solidification shrinkage.

Preliminary SAMPLING of the castings

The production of the new equipment and the process parameters of the first sampling reflect the setup of the virtual simulation. Although the castings produced during the first sampling do not present particular defects on a visual inspection at a rough state, the pressure tightness tests (up to 13Bar in the air) after machining have produced a 30% of production scraps.

Figure 1 - Air-pressure v01
In agreement with the customer, the computerized industrial tomography carried out on the sample has allowed to identify the internal defects caused by air entrapment and/or by isolated volumetric shrinkage, directly in the sample casting and the following computerized elaboration allows its 3D view, as in Figure 3.

The analysis of the results achieved with the tomographies on some samples has allowed to identify two areas of the casting characterized by high concentration of porosity.

The core closeness of such areas to some machined surfaces has generated a high percentage of scrap in presence of porosity. The interpretation of the results achieved with the virtual analysis has therefore permitted the identification of the threshold over which porosity due to air entrapment can be detected as a consequence of the filling dynamics.

The virtual analysis and the tomography have therefore allowed an in-depth understanding of the causes related to the non-quality at the end of the productive cycle but permitted to identify the problem solution.

**OPTIMIZATION of the process**

The approach followed by Tecnopress S.p.A. designers, looking for a solution to improve the process performance, has used the numerical optimization technology available in the MAGMAfrontier module. Once the virtual model is representative of the phenomenon under investigation and reliable in relation to the reference simulation, the optimizer allows a rational investigation on the quantitative effects caused by the variation of the parameters ruling the process and enabling to automatize the detection of the best configurations also for the systems characterized by a high level of complexity, as the die-casting one.

The real sampling activity has allowed to validate by comparison both the representativeness and reliability of the set virtual model. On such model several variables have been identified by technicians, according to their experience, that could be modified to rule the process.

Practically speaking, MAGMAfrontier is a tool requiring the definition of the objectives to be reached, as air entrapment minimization, shrinkage solidification reduction, as highlighted by the tomography analysis, in relation to the process parameters variability and to the casting system geometry. In particular, the geometric configuration of the casting system has been parameterized with respect to the thickness of the casting ingates and to their location on the figure cavity (Figure 4), so to evaluate the effect of filling dynamics on the objectives to be achieved.

Furthermore it has been possible to limit the variability of the input (piston speed) and the output parameters (air breathers or pieces), that according to experience are not considered as possible, as for instance the maximum average speed at the casting ingate, which generates a quick wear of the matrix when overcoming a threshold value.

The optimization phase is based on the first series of configurations to be simulated (DOE, Design of Experiment), that have been selected using the “Reduced Factorial” algorithm (such algorithm is based on two different levels of the “Full Factorial” algorithm, but only a part of the design is generated), together with “Sobol” which covers the vectorial space of the intermediate values in the most efficient way possible. The combination of these algorithms has allowed to fully cover the vectorial space of the input variables, thus guaranteeing their representativeness with respect to the population of the complete design. The following generations have been created by a genetic algorithm, called MDOA, that allows to determine the further design populations by combining the best design of the first DOE sequence using elitism and mutation criteria.

A set of 192 analyzed configurations constitutes the basis for the user to choose the best solution. The scatter chart shows in Figure 5 how each design is associated to a point in the Cartesian diagram, where the axes represent the objectives to be optimized (Figure 5). The best configuration identified by the computer optimizer (Figure 6) has highlighted that the trend to entrap air in the critical zones of the casting is particular sensitive to conformation in the casting ingates. In particular, with respect to the first configuration considered, the first of the three casting ingates had to be adequately thickened, the second needed to be thickened and relocated, whereas the third only was to be relocated. The optimizer therefore suggested a peculiar and not so intuitive configuration; in this sense it has been possible to understand how useful the tool can be, providing an important support to decision making, especially when highly complex problems have to be faced and several choices, not so easy to be compared, have to be evaluated.

Matrices have been coherently modified in agreement with the optimized configuration and the result of the sampling has been definitely satisfying, since it has allowed to reduce the production scrap from the initial 30% to a physiologic 2%. X-Ray controls on the sampled casting with the new industrialized equipment do not highlight porosity unconformity (Figure 7 and Figure 8).

Luca Bracchi and Cristian Bettinsoli, expert designers of casting equipment of Tecnopress S.p.A. have noticed that: “Not only the simulation results match with what we found out during the new equipment sampling campaigns, but also we could take advantage from the use a really useful tool supporting the decision –making process” and they add: “ Tecnopress consider Customer’s satisfaction a decisive factor to keep the market leadership. Proposing good solutions in a short time, without affecting the product cost, makes our customers’ trust grow, with a positive impact on the number of orders”.

**Conclusions**

This work presents a typical case of industrialization of the die-casting production process for an automotive component. The problems identified during the development phase of the new equipment have allowed to prove the advantages offered by the virtual optimization, as support to the decision-making process in a context characterized by a high level of complexity.

In particular, the initial phase of the industrialization has been described; due to the very short time available, despite the advanced design techniques in use, the process was implemented with non-satisfactory qualitative performances, but the following virtual optimization phase has allowed a quick and cheap identification of a solving configuration.

The case under investigation can be considered as general valid demonstration of MAGMAfrontier usefulness to support the designer when looking for a robust and reliable solution to extend the die-casting application field to other sectors, requiring a high level of performance, thus keeping an eye on economic affordability.

**Tecnopress**

Since 1963 Tecnopress is the leader in aluminum die-casting and high-quality machining and finishing.

The products are 100% Made in Italy: Tecnopress strictly monitors the quality of the casting and processes in order to offer the Italian as well as the international markets high-value products.

The strength and key-factor of the company lies in the “dynamic” cooperation with its Customers from the technical feasibility assessment to the integrated supply of the finished castings.

Our target? We want to grow to satisfy and exceed our Customers’ expectations.

Luca Bracchi, Cristian Bettinsoli, Tecnopress Giampietro Scarpa, Nicola Gramegna, EnginSoft
Structural analysis of Microchannel Condenser Heat Exchangers using Finite Element Techniques

Condenser heat exchangers are devices used in air conditioning systems for transferring heat from the hot refrigerant flowing through it to the cold ambient. A microchannel heat exchanger, having higher efficiency than the fin and tube, finds its application in recent times largely on the condenser side of HVAC applications. It consists of header tubes, microchannel tubes, and fins attached to them externally. Structural qualification of the HVAC units requires the unit to pass through a set of thermal conditions during operation and dynamic loads during its transportation. The challenge that a microchannel model can pose for simulation is capturing its complex geometry which comprises of microchannels with minute ports and a large number of slit fins with louvers. Building a model of this complexity results in a large sized model and its simulation calls for large computational time and high end systems. This article presents the methodology of developing an equivalent simplified model of the fins and tubes of the microchannel using an equivalent stiﬀness technique. The model built can be used in studies to predict the impact of the microchannel on its supporting structures, and thermal stresses within the heat exchanger, while approximating the accuracy of a fully detailed analysis.

Introduction

Companies providing air conditioning solutions face the hurdle of constantly providing efficient and reliable equipment. This goal is difﬁcult to achieve when the requirement is to simulaneously retain equipment size and limit potential cost impact. A condenser micro channel is one such solution that is experiencing an accelerated use in air conditioning, both in residential and commercial applications. The reason is mainly due to its compact size and high efficient heat transfer rate compared to the conventional fin and tube heat exchanger. Typically, a microchannel comprises of three components: Flat microchannel tubes which have minute ports through which high pressure, high temperature refrigerant passes, louvered fins connecting the alternate layers of microchannel tubes, which assist in heat transfer from the hot refrigerant to the cool ambient, header tubes on either end which guide refrigerant flow from the inlets of the microchannel tubes to the outlets.

Problem Statement

Structural qualification of the HVAC units requires the unit to pass through a set of thermal conditions during operation and dynamic loads during its transportation. An FE simulation of these cases is carried out on the units considering all its internal components, which ensures the eﬀect of each component on its supporting structure. Due to the intricate geometry of the microchannel coil, building its FE model is a challenging task. The microchannel tubes with its minute ports and the series of slit fins with louvers could be captured only with ﬁne element size that would result in a large FE model of the coil. Contact deﬁnition between the fins and the surface of the tubes would lead to the addition of large number of contact elements. The model thus built will be huge and would require large investment in terms of time for modeling and simulation run-time and would call for a high end computational system.

Method followed in developing the equivalent model

The new developed method reduces the complexity in the problem by replacing the fins and tubes by individual solid blocks whose mass and material stiﬀness properties are tuned, such that they behave like the actual fins and tubes. Fins are very thin metal formed into a bellows or accordion like shape as shown in Fig. 2. If the metal is considered as a solid of the same overall dimensions, the elastic properties are non-isotropic, the stiﬀness is very diﬀerent along the direction of the bellows. The behavior of the fin can be accurately modeled as a solid with orthotropic properties consisting of elastic moduli, shear moduli, and Poisson’s ratios. The approach is accurate for tension, bending, and shear in all planes. The orthotropic solid elastic approach does not work for the microchannel tubes. Bending stiﬀness of a solid equivalent tube will be diﬀerent about the two axes, but there is a single modulus of elasticity in the axial direction to govern it. The microchannel tubes are modeled as an equivalent isotropic solid, but the dominant deformation mode (tension, strong axis bending, weak axis bending) must be determined for the given loading and modulus determined to match.

Density Approximation

A solid block equivalent in dimension to the microchannel tube is modeled. The mass of the original tube is known, M. The volume of the equivalent solid block can be determined, Vequi.

Thus the density of the solid block to weigh same as the original is estimated as

Densityequi = M1/Vequi

Moduli Prediction

Tubes: Unit load is applied on the microchannel tube along a given axis and its deformation is calculated. For the unit load, the modulus required to produce the same deformation in the equivalent model is calculated directly by a simple ratio of the deformations of the two models. The predicted modulus corresponds to the stiﬀness value about that loading direction of the tube, generally weak axis bending, strong axis bending, or tension.

Fins: Two adjacent microchannel tubes and its intermediate fins are considered, similarly another model wherein the fins are replaced by an equivalent solid block is modeled. Both the models are subjected to tension and shear loadings in three directions, as shown in the fig.3. Each of these load cases has an orthotropic modulus that dominates the deformation, but all moduli have some eﬀect on all deformations. Manual iteration was used to ﬁnd the orthotropic modulus matrix that produced accurate results for all load cases. After the initial guess, a simple ratio of deformations for each case was used to predict the next value of the dominant modulus for that case, as shown in the table 1.

After several iterations, the ﬁnal matrix was arrived at, as shown in the ﬁnal solution table 2. The Poisson’s ratios for two directions needed to set to very small values to get accurate results and to make the orthotropic property matrix deﬁnite.

This is method is followed to model the equivalent solid blocks that represent the fins and tubes.
The strong axis equivalent modulus for the tube was used in this case.

The coil slab, which corresponds to the strong axis bending of each tube.

Figure 6. These modes are dominated by bending about the weak axis of the coil and its influence on the supporting structure is as shown in

The modal response due to road vibration input is an important consideration. The thermal profile after steady state analysis and stress generated due to temperature is for the transport HVAC unit studied.

An analysis was undertaken for a transport HVAC unit, where dynamic methodology.

Validation of this methodology with actual prototype build of this transportation unit has been planned to include both thermal and dynamic tests. The outcome of these tests will provide much more confidence in

The use of this methodology in developing the condenser coil or any similar components/structures can be validated by comparison of complete system response against test results. The modal and thermal response obtained for the complete assembly by the use of simplified model captures the behavior of system in application. Hence the new methodology is beneficial in modeling the complex system without compromising the accuracy of the results.

The composite sandwich laminate

For this study, 3 samples of composite sandwich laminates have been kindly provided by Benecor Inc. The sandwich laminate samples have carbon fiber composite faces and a core made of commercially pure titanium (TiCP). The faces are made of 9 unidirectional fiber plies in an epoxy matrix, with a [0°90°0°90°0°90°0°90°0°] ply book. The core is made of square cells welded at the intersection corners. A sample of the provided sandwich is shown in figure 1.

Table 1 - Equivalent stiffness calculation – example iteration

<table>
<thead>
<tr>
<th>Load case</th>
<th>modulus of reaction</th>
<th>modulus of deformation</th>
<th>modulus of rigidity</th>
<th>modulus of inertia</th>
<th>modulus of compliance</th>
<th>modulus of compliance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial tension</td>
<td>Ex</td>
<td>835.71</td>
<td>856.8</td>
<td>25372.58</td>
<td>0.9987</td>
<td>25346.52</td>
</tr>
<tr>
<td>Shear only</td>
<td>Gxy</td>
<td>31.76</td>
<td>31.76</td>
<td>312.47</td>
<td>1.0000</td>
<td>312.47</td>
</tr>
</tbody>
</table>

Table 2 - Equivalent stiffness calculation – final solution

<table>
<thead>
<tr>
<th>Load case</th>
<th>modulus of reaction</th>
<th>modulus of deformation</th>
<th>modulus of rigidity</th>
<th>modulus of inertia</th>
<th>modulus of compliance</th>
<th>modulus of compliance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial tension</td>
<td>Ex</td>
<td>835.71</td>
<td>856.8</td>
<td>25372.58</td>
<td>0.9987</td>
<td>25346.52</td>
</tr>
<tr>
<td>Shear only</td>
<td>Gxy</td>
<td>31.76</td>
<td>31.76</td>
<td>312.47</td>
<td>1.0000</td>
<td>312.47</td>
</tr>
</tbody>
</table>

Figure 5 - Microchannel assembly with equivalent solid blocks

Figure 6 - Modal response of the unit inclusive of the condenser coil

FE modeling of a composite sandwich laminate with LS-DYNA for Aerospace applications

Laminated composite materials such as sandwich structures are widely used in the transportation industry due to their high stiffness and strength to weight ratio. Significant efforts have been made in improving the structural theories and prediction methods to give engineers a reliable tool to develop new structural concepts (i.e. vehicle front and rear end components, automotive and aeronautical seats). The complete mechanical behavior of a composite material is very difficult to be predicted without first calibrating the model to some test data. This is especially important when damage and rupture are considered for impact and crash analyses.

This project investigates how best to correctly predict the energy absorption of these non-isotropic and non-homogeneous materials, whose lamina thickness is typically of the order of 0.2 mm or less and consists of tiny fibers or particles dispersed in a matrix material. This makes it impossible to model details in the micro scale when considering a whole component or vehicle. The typical FE element size for metal structures is 5mm and needs to be similar for composite structures to ensure manageable simulation times.

Therefore, in order to achieve computational efficiency relevant models and theories need to be understood. This article examines some modelling techniques to simulate a sandwich composite laminate, paying particular attention to its non-linear behavior and correlated energy absorption features. The numerical results have been compared with experimental tests.

The composite sandwich laminate

In this study, 3 samples of composite sandwich laminates have been kindly provided by Benecor Inc. The sandwich laminate samples have carbon fiber composite faces and a core made of commercially pure titanium (TiCP). The faces are made of 9 unidirectional fiber plies in an epoxy matrix, with a [0°90°0°90°0°90°0°90°0°] ply book. The core is made of square cells welded at the intersection corners. A sample of the provided sandwich is shown in figure 1.
The geometrical properties of the sandwich samples are summarized in Table 1. The density of the TIP core is 11.7 pcf (187.4 Kg/m³).

The mechanical properties of the materials (carbon-epoxy lamina and DP titanium) were not available.

Reasonable values for the first analyses were obtained from literature and then a combination of experimental tests and trial FE analyses were used to obtain more reliable material data.

### Mathematical modeling strategy

Many models are available that simulate the mechanical behavior of a sandwich laminate: their degree of complexity and accuracy depends on what behavior the model was designed to assess. In general, it is advisable to use the simplest possible model to predict the desired phenomena.

For example, if the aim is to predict linear stiffness and displacements, critical buckling load or early natural modes, a simple model of the sandwich laminate should be used. Conversely, if the aim is to predict the behavior of the laminate under non-linear combined loads, which include large and plastic deformation or stresses and strains normal to the mid-plane, a FSDT plate model that accounts for a correct shear factor is required. All these materials require equivalent non-linear curves to define the constitutive equations under pure loads (e.g. pure shear or pure axial load), and can implement other parameters such as failure criteria and dumping for a more accurate simulation.

The chosen model is MAT 142 because it is relatively simple, requires few input curves and does not suffer from non-linear deformations due to the decoupling of curves, as encountered for example in the case of crushable foam. All these models are therefore of utmost importance in explicit FE crash simulations. The model for the core is MAT 142 because it is relatively simple, while the model for the skins is MAT 125 (reinforced foam) and MAT 142 (transversely anisotropic foam). All these models are therefore of utmost importance in explicit FE crash simulations.

The relevant material models for composite skins in LS-DYNA are MAT 22, 54, 55, 58, 59, 104, 108, 114, 116, 117, 118, 158, 161, 162. The large number of materials shows the huge effort that has to be made to create models that satisfactorily predict the behavior of the laminate.

In addition to the LS-DYNA manuals, it is always helpful to refer to relevant technical papers, run some benchmark simulations, and compare the results with hand calculations or known results. This study incorporates all these methodological aspects for a sandwich laminate.

Table 1 - Geometrical sample sizes of the laminate

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width</td>
<td>50.80</td>
</tr>
<tr>
<td>Height</td>
<td>6.00</td>
</tr>
</tbody>
</table>

The model for the skins

One important decision is to choose the right material model for the skin laminate. LS-DYNA has a wide range of material models, with a number of materials specific for composite laminates. For nonlinear analyses, it is important to the materials to include plasticity, stiffness degradation and failure. For dynamic analyses, it is also advisable to use a correct value of damping. All these properties influence the energy absorption of the structure, and are therefore of utmost importance in explicit FE crash simulations.

The relevant material models for composite shells in LS-DYNA are MAT 22, 54, 55, 58, 59, 104, 108, 114, 116, 117, 118, 158, 161, 162. The large number of materials shows the huge effort that has to be made to create models that satisfactorily predict the behavior of the laminate.

In addition to the LS-DYNA manuals, it is always helpful to refer to relevant technical papers, run some benchmark simulations, and compare the results with hand calculations or known results. This study incorporates all these methodological aspects for a sandwich laminate.

For this study, the aim is to assess the energy absorption of the laminate, therefore a good compromise between model detail / accuracy and simulation time is a shell-brick-shell model (the core modeled as a solid between two shell layers). The upper and lower layers can be effectively modeled with an equivalent FSDT plate or a higher order theory, with possibly stress recovery for augmented accuracy.

The main advantage of this modelling method is that it can predict better the kinematic behavior under non-linear combined loads, which include large and plastic deformation or stresses and strains normal to the mid-plane. The shell-brick-shell model can also be used to model failure modes (that can be many and complex for a sandwich laminate), such as skin failures (e.g. face yielding or face wrinkling), core failures (shear or indentation), or the adhesive failure between the core and outer layers. This can be done using a TIE_BREAT contact definition in LS-DYNA, but has not been studied in this paper. All these features are important in correctly predicting the energy absorption of the sandwich laminate.

The model for the core

In order to minimize the computation time for the largest problems, simplified models are usually necessary. This is the case for sandwich structures with honeycomb cores, which are made of a dense grid of small plates whose detailed modelling would be computationally very expensive, especially in an explicit analysis where the minimum dimension of the elements determines the numerical stability and time step of the whole model.

It is therefore important to use a simplified model for the core. There are many equivalent material models for these cores in LS-DYNA, to be used with solid elements. Among them there are MAT 63 (crushable foam), MAT 26 (honeycomb), MAT 126 (modified honeycomb), and MAT 142 (transversely anisotropic foam). All these materials require equivalent non-linear curves to define the constitutive equations under pure loads (e.g. pure shear or pure axial load), and can implement other parameters such as failure criteria and dumping for a more accurate simulation.

The chosen model is MAT 142 because it is relatively simple, requires few input curves and does not suffer from non-linear deformations due to the decoupling of curves, as encountered for example in the case of crushable foam. All these models are therefore of utmost importance in explicit FE crash simulations.

For the axial crush test a more refined model is required, as it is important to use the simplest model for the core. There are many equivalent material models for these cores in LS-DYNA, to be used with solid elements. Among them there are MAT 63 (crushable foam), MAT 26 (honeycomb), MAT 126 (modified honeycomb), and MAT 142 (transversely anisotropic foam). All these materials require equivalent non-linear curves to define the constitutive equations under pure loads (e.g. pure shear or pure axial load), and can implement other parameters such as failure criteria and dumping for a more accurate simulation.

The chosen model is MAT 142 because it is relatively simple, requires few input curves and does not suffer from non-linear deformations due to the decoupling of curves, as encountered for example in the case of crushable foam. All these models are therefore of utmost importance in explicit FE crash simulations.

For the axial crush test a more refined model is required, as it is important to use the simplest model for the core. There are many equivalent material models for these cores in LS-DYNA, to be used with solid elements. Among them there are MAT 63 (crushable foam), MAT 26 (honeycomb), MAT 126 (modified honeycomb), and MAT 142 (transversely anisotropic foam). All these materials require equivalent non-linear curves to define the constitutive equations under pure loads (e.g. pure shear or pure axial load), and can implement other parameters such as failure criteria and dumping for a more accurate simulation.

The chosen model is MAT 142 because it is relatively simple, requires few input curves and does not suffer from non-linear deformations due to the decoupling of curves, as encountered for example in the case of crushable foam. All these models are therefore of utmost importance in explicit FE crash simulations.

For the axial crush test a more refined model is required, as it is important to use the simplest model for the core. There are many equivalent material models for these cores in LS-DYNA, to be used with solid elements. Among them there are MAT 63 (crushable foam), MAT 26 (honeycomb), MAT 126 (modified honeycomb), and MAT 142 (transversely anisotropic foam). All these materials require equivalent non-linear curves to define the constitutive equations under pure loads (e.g. pure shear or pure axial load), and can implement other parameters such as failure criteria and dumping for a more accurate simulation.
In fact, for a sandwich beam loaded at the mid-point in 3 point ASTM standard beam theory, modified for a sandwich beam according to the possible using some hand calculations based on the Timoshenko formula. From the experimental test results, it is possible to extract other have been obtained; they are shown in figures 7 and 8. From these tests the force-displacement curves of the laminates deformation was sufficient to produce a failure strain.

Wheatstone bridge to obtain a measure of the strain gauges have then been linked to a Comazzi 20T, where the sample rests on two steel cylinders, 140mm apart from each other. The load is applied through a third cylinder with a diameter of 25 mm, put at the middle span of the laminate. The load is applied manually and the displacement at the middle cylinder is measured by two LVDT displacement transducers, situated on the two extremities of the section at the middle span of the laminate. Their values are then averaged in order to gain a more precise value of the displacement at the middle section. In order to obtain further information from the tests, a strain gauge was placed on the center of the compressed face of each sandwich laminate. The strain gauges have then been linked to a Wheatstone bridge to obtain a measure of the strain. The environment during the tests was monitored; the room temperature was 22° Celsius with a humidity of 52%. The samples were deflected by about 4 mm, which were the maximum stroke of the displacement transducers. However, this deformation was sufficient to produce a failure mode. From these tests the force-displacement curves of the laminates have been obtained, they are shown in figures 7 and 8. From the experimental test results, it is possible to extract other interesting data to compare to the numerical model. This is possible using some hand calculations based on the Timoshenko beam theory, modified for a sandwich beam according to the ASTM standard. In fact, for a sandwich beam loaded at the mid-point in 3 point flexure, it is possible to obtain the equivalent shear modulus G of the core as where d is the total thickness of the sandwich beam, c is the thickness of only the core, b is the width, and U is the shear stiffness of the whole laminate. This can in turn be represented as:

\[ G = \frac{4cU}{(d + c)^2b} \]

where \( W \) is the maximum displacement on the middle point, \( P \) is the applied load, \( L \) is the useful length, and \( D \) is the flexural stiffness. This last term could not have been obtained by direct calculation from the data using the elastic modulus \( E \) of the faces, because these were not available from the manufacturer. So it has been obtained using the data gathered by the strain gauge, because from assumptions it follows that \( M = D \frac{\partial \varepsilon}{\partial x} \) with

\[ \varepsilon = \frac{2x}{W} \]

so that

\[ D = \frac{M}{2x} \]

where \( \varepsilon \) is longitudinal to the fibers where averaged because these were not available from the manufacturer. So it was found that the stress-strain curve of titanium taken from literature was of inferior mechanical properties when compared to the actual material of the core, and was therefore consequently recalibrated. If actual material properties had been available, the final curve would probably fit better the experimental curves.

Conclusions

The non-linear flexural behavior of a composite sandwich laminate has been investigated, so that the energy absorbed by the structure could be evaluated, by integrating the resulting force-displacement curve. The force-displacement curve from the FE model set with LS-DYNA shows a good agreement with experimental data (figure 11). The biggest source of error is the lack of mechanical data of the original materials. Table 4 summarizes the results of the experimental data and compares them with data from the FE model. Analytical results of the cellular core properties are also added for comparison.
Numerical simulations are the scientific tool used to predict a physical phenomenon without empiric observations. The underlying model is the link between the real environment and the virtual settings, thus its accuracy influences the reliability of the entire simulation.

Even if the model can be refined ensuring better results, a certain lack of knowledge could characterize it through uncertainties of the parameters involved. These uncertainties can be due to intrinsic volatility or incomplete knowledge of the quantities considered. This situation is not so far from reality: values of physical measures are often given as confidence intervals and in certain cases they are not even exactly known. Therefore a single deterministic simulation may not be enough to describe the physical phenomenon, even if the model equations are deterministic.

For all the above reasons Uncertain Quantification (UQ) methods are developed, where the uncertainties are taken into account in simulations. UQ is a fundamental step towards validation and certification of numerical methods used for critical decisions in presence of uncertainties. Validation consists in comparing the simulated results with empiric measurements, performed on real systems, in order to achieve good prediction of reality.

Figure 1 - Magnitude of the difference between deterministic simulation and NISP approach (a UQ method) for an advection problem at first time step $t = 0.1$ (left) and last time step $t = 1$ (right).

Figure 2 - Log-Log scale plot of the absolute error for the mean of a process computed with PCE, MC, QMC and LHS techniques.

Uncertainties can be classified as: epistemic or aleatory. The former is a potential deficiency, due to a lack of knowledge. It could be a consequence of assumptions introduced in the derivation of the mathematical model, such as limited accuracy in the measurement of the physical constant involved (it is also called reducible uncertainty or incertitude). Such kind of uncertainty source can be reduced by redefining the model and increasing its accuracy.

The latter is the intrinsic physical variability of the system. It is

Itacae is a consulting company from Turin that collaborates with EnginSoft in many projects regarding explicit dynamics using LS-DYNA code. The company has staff with extensive experience in the best-in-class methodologies for the analysis of industrial products & manufacturing processes, for the optimum management of the product development cycle. The main customers are manufacturers & design offices in the sectors of automotive & aerospace industry.

Application to an aircraft seat

The laminate in question has been applied to an aircraft seat, in order to investigate possibility of reducing the seat’s mass, while improving its structural performance in crashworthiness.

Using this advanced material allows the weight of the seat back to be reduced from 2.8 Kg to 1.4 Kg.

Acknowledgements

With great thanks to Benecorinc., Wichita (KS), for providing samples of sandwich laminates for the experimental tests.

Alessandro Tazzini - ITACAE srl
Dario D’Agostino - EnginSoft
Marco Di Sciara, Massimiliano Mattone
Politecnico di Torino

Itacae is a consulting company from Turin that collaborates with EnginSoft in many projects regarding explicit dynamics using LS-DYNA code. The company has staff with extensive experience in the best-in-class methodologies for the analysis of industrial products & manufacturing processes, for the optimum management of the product development cycle. The main customers are manufacturers & design offices in the sectors of automotive & aerospace industry.

Table 4 - Summary and comparison of results

<table>
<thead>
<tr>
<th>Material</th>
<th>Analytical</th>
<th>FE model</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>E11 [GPa]</td>
<td>4.36</td>
<td>4.4</td>
<td>-</td>
</tr>
<tr>
<td>E22=E33 [Gpa]</td>
<td>0.367</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>G12=E013 [Gpa]</td>
<td>0.75×1.44</td>
<td>0.894</td>
<td>0.891</td>
</tr>
<tr>
<td>G23 [Gpa]</td>
<td>0.74×1.40</td>
<td>0.9740</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 3 - Contour plot of strain energy distribution

Figure 4 - Force-displacement plot of the FE model

Figure 5 - Crach test on the aircraft seat, modified using the studied composite laminate

Table 4 - Summary and comparison of results

<table>
<thead>
<tr>
<th>Material</th>
<th>Analytical</th>
<th>FE model</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>E11 [GPa]</td>
<td>4.36</td>
<td>4.4</td>
<td>-</td>
</tr>
<tr>
<td>E22=E33 [Gpa]</td>
<td>0.367</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>G12=E013 [Gpa]</td>
<td>0.75×1.44</td>
<td>0.894</td>
<td>0.891</td>
</tr>
<tr>
<td>G23 [Gpa]</td>
<td>0.74×1.40</td>
<td>0.9740</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 11 - Crash test on the aircraft seat, modified using the studied composite laminate

Figure 12 - Force-displacement plot of the FE model

Numerical simulations are the scientific tool used to predict a physical phenomenon without empiric observations. The underlying model is the link between the real environment and the virtual settings, thus its accuracy influences the reliability of the entire simulation.

Even if the model can be refined ensuring better results, a certain lack of knowledge could characterize it through uncertainties of the parameters involved. These uncertainties can be due to intrinsic volatility or incomplete knowledge of the quantities considered. This situation is not so far from reality: values of physical measures are often given as confidence intervals and in certain cases they are not even exactly known. Therefore a single deterministic simulation may not be enough to describe the physical phenomenon, even if the model equations are deterministic.

For all the above reasons Uncertain Quantification (UQ) methods are developed, where the uncertainties are taken into account in simulations. UQ is a fundamental step towards validation and certification of numerical methods used for critical decisions in presence of uncertainties. Validation consists in comparing the simulated results with empiric measurements, performed on real systems, in order to achieve good prediction of reality.

In Figure 1 the magnitude of the difference between a deterministic simulation of a process and a simulation, using a UQ method, is pointed out, displaying how input uncertainties influence the output of a simulation.

Uncertainties can be classified as: epistemic or aleatory. The former is a potential deficiency, due to a lack of knowledge. It could be a consequence of assumptions introduced in the derivation of the model equations. The latter is the intrinsic physical variability of the system. It is

Table 4 - Summary and comparison of results

<table>
<thead>
<tr>
<th>Material</th>
<th>Analytical</th>
<th>FE model</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>E11 [GPa]</td>
<td>4.36</td>
<td>4.4</td>
<td>-</td>
</tr>
<tr>
<td>E22=E33 [Gpa]</td>
<td>0.367</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>G12=E013 [Gpa]</td>
<td>0.75×1.44</td>
<td>0.894</td>
<td>0.891</td>
</tr>
<tr>
<td>G23 [Gpa]</td>
<td>0.74×1.40</td>
<td>0.9740</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 11 - Crash test on the aircraft seat, modified using the studied composite laminate

Figure 12 - Force-displacement plot of the FE model

Table 4 - Summary and comparison of results

<table>
<thead>
<tr>
<th>Material</th>
<th>Analytical</th>
<th>FE model</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>E11 [GPa]</td>
<td>4.36</td>
<td>4.4</td>
<td>-</td>
</tr>
<tr>
<td>E22=E33 [Gpa]</td>
<td>0.367</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>G12=E013 [Gpa]</td>
<td>0.75×1.44</td>
<td>0.894</td>
<td>0.891</td>
</tr>
<tr>
<td>G23 [Gpa]</td>
<td>0.74×1.40</td>
<td>0.9740</td>
<td>-</td>
</tr>
</tbody>
</table>

Figure 11 - Crash test on the aircraft seat, modified using the studied composite laminate

Figure 12 - Force-displacement plot of the FE model

Table 4 - Summary and comparison of results

<table>
<thead>
<tr>
<th>Material</th>
<th>Analytical</th>
<th>FE model</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>E11 [GPa]</td>
<td>4.36</td>
<td>4.4</td>
<td>-</td>
</tr>
<tr>
<td>E22=E33 [Gpa]</td>
<td>0.367</td>
<td>1.14</td>
<td>-</td>
</tr>
<tr>
<td>G12=E013 [Gpa]</td>
<td>0.75×1.44</td>
<td>0.894</td>
<td>0.891</td>
</tr>
<tr>
<td>G23 [Gpa]</td>
<td>0.74×1.40</td>
<td>0.9740</td>
<td>-</td>
</tr>
</tbody>
</table>
not linked with lack of knowledge and cannot be reduced (also referred as variability, stochastic uncertainty).

Aleatory uncertainties motivate the introduction of UQ methods. Among them the most famous one is Monte-Carlo (MC), that relies on pseudo-random sampling of inputs parameter of a process. The model is simulated on these realizations, and the results allow to compute statistics, probability law, etc., of the output. MC is a robust method, but it suffers from low convergence rate: many simulations are needed for good approximation. Moreover, MC is cumbersome and quite involved when the process requires a lot of time in detecting the output.

In order to decrease the computational costs of MC simulations the Polynomial Chaos Expansion (PCE) approach is developed. The first advantage relies on approximating the quantity of interest as a linear combination of polynomials, which are efficiently implemented by software for numerical computation. The PCE is detected by means of Non-Intrusive Spectral Projection (NISP) technique which exploits a collection of deterministic simulations of the process. Thus the deterministic model is not recast into a probabilistic framework: it lives at it stands. Moreover, to get the same precision as MC less simulations are required, saving computational costs. Such features are shown in Figure 2 and Figure 3, where the absolute error of the mean and variance of a certain process, computed with PCE, are compared with the errors resulting from MC simulations and some improved MC methods, such as Quasi-Monte-Carlo (QMC), Latin Hypercube Sampling (LHS).

The NISP technique was studied by Gregorio Pellegrini during an internship at EnginSoft, under the supervision of Manolo Venturini PhD. Moreover, this was the starting point for investigating the theoretical properties of PCE and features of NISP approach, which were the core of Gregorio Pellegrini’s thesis “Polynomial Chaos Expansion with applications to PDEs” (advisor Marco Caliari PhD) for the Master’s Degree in Mathematics at University of Verona. During the internship Scilab software was employed, since its toolbox NISP collects all routines for computing either the univariate or multivariate PCE (via Non-Intrusive approach). Moreover, the user can customize several features of the decomposition and deal with post-processing analysis, such as computations of statistics of the output, samplings and sensitive analysis.

In Figure 4 and Figure 5 some post-processing features of an application of NISP to the lid-driven cavity problem are shown (where the lid velocity is the uncertain parameter): a PCE of a cross-section of the velocity field is computed displaying its mean and variance. The data set is detected using FreeFem++: a Partial Differential Equation (PDE) solver based on the Finite Element Method.

Gregorio Pellegrini, University of Verona

Advanced Python Scripting in Kraken - Well Placement Optimization Using Genetic Algorithms

When developing exploration strategy for an oil field or when evaluating the technical and economic aspects of an exploitation project, reservoir engineers have to use all information previously gathered in seismic acquisition, wireline profiling, correlation wells as well as known information of other reservoirs in the same basin to try to characterize and model the reservoir as best as possible.

The exploration strategy will depend on the reservoir model since it will be used to perform simulations of multiple production scenarios and uncertainty analysis. At this stage, the reservoir engineer tries to decide the recovery strategy, which will depend on the content of reservoir fluids: primary recovery for gas fields, water or gas injection for oil fields, steam injection for heavy oil and many others techniques for different types of reservoirs and fluids.

To highlight this work, a deep water oil field with an aquifer and an uneven permeability field, as showed in the Figure at the top, requires development. The engineering decisions should be the number of production wells, their geometry (vertical, horizontal or directional) and their location, the number of injection wells and the injection strategy (injectors around the producers or in a grid).

The definition of the geometry was done based on the permeability distribution which did not present any preferential direction that might be explored by a directional or horizontal wells, so leading to a decision for vertical wells, which are simpler and cheaper. The actual number and location of the producers and injectors wells then became the problem to be solved. For this, many scenarios should be simulated to obtain the maximum Net Present Value and then provide this financial report to the economic analysis.

However, simulating several scenarios based only in the observations/recommendations of the reservoir engineer will not guarantee that a certain configuration is the optimum or not even close to the best scenario, at least not in a feasible project time. However, an automated process combined with an optimization algorithm could provide a much better result in a short time, since it could run several simulations and automatically change well positions according to the optimization algorithm. Therefore, a software that is able to read reservoir simulation result files and optimization algorithms was required to perform this task and the ESSS Kraken software programme was selected due to its ability to handle simulation files and the presence of its powerful Python Scripting interface, which allows the usage of optimization algorithms available in the Python community repositories.

Five scenarios with different numbers of producers and injectors wells were created, in which the genetic algorithm then varied the location of each well in order to obtain the maximum NPV for each configuration. The producer wells could be placed only in the oil region and the injectors could be placed anywhere in the reservoir.

The studied scenarios are detailed in Table 1 as well the results of each configuration. It can be seen that the highest NPV is the obtained in the configuration number 2, with 8 producers and 7 injectors, even though this configuration does not present the highest recovery factor. This happens because the NET present values considers not only the oil production, but also when the oil is produced, favoring the scenario where larger amounts of oil are produced in the first years. The resulting location of the 8 producers and 7 injectors wells are shown in Figure 1.

<table>
<thead>
<tr>
<th>Scenarios</th>
<th>Number of Producers</th>
<th>Number of Injectors</th>
<th>Recovery Factor</th>
<th>Net Present Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8</td>
<td>6</td>
<td>35,73%</td>
<td>1.403 MM</td>
</tr>
<tr>
<td>2</td>
<td>8</td>
<td>7</td>
<td>30,29%</td>
<td>1.512 MM</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>7</td>
<td>30,54%</td>
<td>1.366 MM</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>8</td>
<td>30,02%</td>
<td>1.117 MM</td>
</tr>
<tr>
<td>5</td>
<td>12</td>
<td>8</td>
<td>30,04%</td>
<td>824 MM</td>
</tr>
</tbody>
</table>

Table 1: Studied scenarios

Figure 7: Resulting well placement

Marcus Reis - ESSS Brazil
Multi-Body Dynamics for ANSYS

You love ANSYS! But have you ever wanted access to a strong Multi-Body Dynamics (MBD) capability from within your Workbench user environment? It would be great to be able to simulate the motion and all of the related physics of your mechanical assemblies. You could understand the behavior of your assembly up-front and you could generate the loads that you need for your component-level FEA. A strong MBD capability in Workbench would need to model contacts easily and solve assemblies with contacts quickly. It would let the user define a set of load cases by selecting output points of the work cycle simulation, and automatically define the linear statics models for the component of interest. It would provide MBD-specific advanced post processing capabilities. And finally, it would provide a way to transfer your MBD model to an advanced MBD tool (without losing the work done) when specialized MBD capabilities are needed.

We support MBD for ANSYS after considering the requirements and potential benefits just listed.

Fully functional MBD software in the Workbench Environment

MBD for ANSYS is tightly integrated into Workbench for MBD system creation and data management.

It is also integrated with ANSYS Mechanical for MBD model development, editing, analysis, and post processing. A selection of the menus is shown in the ANSYS Mechanical figure.

High Performance Contact Modeling

MBD for ANSYS uses RecurDyn solver technology and is able to solve contact problems efficiently. For example, the gear pair shown in the figure has many surfaces on each gear that can contact with any of the surfaces of the other gear. With MBD for ANSYS the gear pair rotating at a high speed for 0.1 seconds is simulated in 12 seconds.

Automated, Intelligent Load Transfer Function

Load transfer consists of these steps:

1. The user runs the MBD simulation.
2. A body is selected.
3. Forces on the body are plotted and the user selects output times in the simulation that have loads of interest.
4. A system for the linear statics analysis of each load case is created in the project schematic window, including data connections.
5. The body loads are mapped to regions on the body geometry.
6. The loads cases are analyzed.

Advanced MBD Post Processing Tool

Output forces can be plotted within ANSYS Mechanical. Animation and advanced plotting is done using the RecurDyn Viewer, which runs at a click on the RecurDyn Viewer command. A new window appears with the model geometry and all results are ready for review. Combined views of animations and plots can be defined. Animations can be saved to an avi file for display in presentations. Curve data can be mathematically processed and filtered. Plot data can be exported to a file that can be imported by Excel.

Export the MBD model to the Standalone RecurDyn Software

If additional MBD functionality is needed, a single command exports the MBD model and the resulting files can be imported into standalone RecurDyn/Professional with a single command. All geometry, joints, forces and contacts are retained. The user can add controllers, belts, chains, and bearings. Users can add flexible media to their models. Track assemblies for construction equipment and military vehicles can be added.

Recurdyn and MBD for ANSYS are supported in Europe by EnginSoft. View the MBD for ANSYS podcast to understand the capabilities: www.vimeo.com/125666635

For more information and for a demo of this technology: Fabiano Maggio, Recurdyn Expert in EnginsSoft f.maggio@enginsoft.it

Software as a Service

The new challenge for scientific computation

In today’s digitally connected world, our ability to take advantage of having services at our fingertips has become the norm with our own personal devices. EnginSoft will soon be announcing some innovative solutions relating to cloud computing that has been in development for the dematerialization of company services. Among the solutions will include the forthcoming release of software on demand services and software as a service solution. This will revolutionize the way company’s access software and will transform the benefits from a fiscal point of view, for cost efficient solutions that can suit several scenarios for users of computation software and suites, especially in the multi-physics sector.

A ‘package of hours’ will be available, to be used by companies with full control and modest budget where they can utilise the hours to assist with unpredictable conditions such as: cover analysis peaks, upgrades, and/or modules that are to be used but there are not available in the licence pool.

The service will therefore allow flexibility among a pool of modules, those of specific interest and those scaled down from the paid working hours, proportional to the module importance. This new solution will be able to cover an increasing demand for functionalities, working peaks, and takes advantage of the fiscal benefits (total deduction of these expenses) with very easy-to-use tools. Stay tuned for more updates or contact: Roberto Maini, EnginSoft: r.maini@enginsoft.it

Software Update

Newsletter EnginSoft Year 12 n°2 - 52

53 - Newsletter EnginSoft Year 12 n°2

Software Update
Advanced IC engine design with ANSYS FORTÉ

ANSYS FORTÉ is a CFD package for realistic 3D modeling of internal combustion engines

The great aim of engine manufacturers worldwide is to produce high efficiency and low-emission engines for the wide variety of fuels available. Understanding how changes in fuels composition or engine design can impact the combustion processes is fundamental.

CFD simulation has the potential to help create high performance engines with reduced emissions. However, simulation needs to account for both the physics and the chemistry involved in the combustion process to obtain accurate and predictive results.

An ANSYS FORTÉ simulation solves the full Reynolds-averaged Navier-Stokes (RANS) equations with well-established flow turbulence models. However, different from other CFD software, ANSYS FORTÉ has an advanced chemistry solver that enables the direct use of reaction mechanisms that contain hundreds of chemical species, but with quick simulation times that are usually associated with mechanisms that are orders of magnitude smaller. These enhancement allow designers to accurately predict ignition, emission, combustion duration and engine performance without compromising the geometric fidelity or solution performance. By combining the advanced chemistry solver and the multi-component fuel vaporization models, it makes possible to simulate realistic fuel surrogates in 3D engine models. ANSYS FORTÉ software also employs advanced models to capture spray physics accurately, predict soot formation and calculate knock intensity. Finally, in addition to the modeling capabilities, the ANSYS FORTÉ user interface provides a guided user experience for setup, simulation, and visualization of results.

Automatic Mesh Generation

A typical CFD analysis begins by defining the computational mesh that will represent the geometry of the system. The construction must account for the fact that the mesh must transform and shift dynamically with the motion of pistons and valves during the engine cycle. ANSYS FORTÉ provides several options for defining the mesh; you can load the geometric surfaces from a CAD STL file and then use the automatic mesh-generation option, or you can load in a predefined body-fitted mesh. For in-cylinder-only engine simulations, you also have the option to use a guided Sector Mesh Generator to generate a body-fitted sector mesh from basic piston-bowl profile information.

From an accuracy point of view, the ideal mesh created is one that is Cartesian, with perfectly orthogonal faces, and one in which the boundary conditions are applied exactly on the physical surfaces of the real geometry. Automatic mesh-generation methods that use a pure Cartesian-based system avoid the problems of highly skewed cells that can be introduced with other approaches.

Advanced Chemistry Solver Module

Designing new, high-efficiency, low-emissions IC engines presents technical challenges that are often dominated by the chemical kinetics that occur during combustion. Consequently, combustion simulations for enhanced engine designs require chemistry that can adequately represent real fuel kinetics.

ANSYS FORTÉ incorporates the Model Fuel Library, a database of detailed chemical mechanisms for over 60 fuel components, which represents every class of reaction important for combustion simulation. Its detailed kinetics and fuel composition models can capture all aspects of modern engine behavior, including fuel ignition, flame propagation, pollutant emissions, particle formation and engine knocking, as well as the effects of fuel variability and multistage detonation.

The use of detailed chemical kinetic mechanisms to describe fuel combustion often introduces a large number of stiff, nonlinear, ordinary differential equations (ODEs) that provide numerical problems and long run times. One of ANSYS FORTÉ’s major strengths is its ability to handle realistic fuel-combustion mechanisms that include large numbers of species and reactions, within simulation times that are practical for design. The CHEMKIN-PRO solver technology employed by ANSYS FORTÉ improves the chemistry solution efficiency. The solution technique includes a dynamic adaptive chemistry (DAC) and a dynamic cell clustering (DCC) method.

The chemistry-solution portion of the species equations are solved on a cell-by-cell basis. As a result, the equations are solved independently from the mass and volume of a specific cell. In this way, cells that have the same temperature, pressure, and initial species mass fractions will yield the same result. To take advantage of this fact, the Dynamic Cell Clustering (DCC) method is used to group computational cells of high similarity into clusters using an efficient data-clustering method that requires the kinetic equations to be solved only once for each cluster. The optimal number of clusters is dynamically determined for each CFD time step. The clustering algorithm is exclusively based on the cells’ thermochromic states and is independent of their locations in the CFD mesh. To ensure validity over a wide range of thermochromic conditions, comprehensive kinetic mechanisms for the combustion of realistic fuels typically include hundreds of species and thousands of elementary reactions. However, much smaller subsets of species and reactions often are adequate to capture the dominant reaction pathways for specific local conditions over a short time span. Typically taken to be the hydrodynamic time scale in CFD calculations. To make use of this fact, the dynamic adaptive chemistry (DAC) method reduces the detailed mechanism to locally valid smaller mechanisms, on the fly. The reduction is dynamically applied cell-by-cell (or cluster-by-cluster if the Dynamic Cell Clustering method is used) at each time step. In this way the dynamic chemistry solution often uses locally valid, smaller mechanisms instead of the full mechanism, causing significant time savings for the overall transient integration. These solution models also make change the established scaling equation: instead of scaling with the cube of the number of species, as is normally the case, the simulation time scales linearly.

Spray Models

Another challenge in engine simulation is in spray modeling. The choice of the spray model in an IC engine can have a significant impact on both time-to-solution and the accuracy of results.

ANSYS FORTÉ includes advanced models to simulate multi-component fuel-spray dynamics and spray interactions with flowing multi-component gases. The modeled sub-processes include nozzle flow, spray atomization, droplet breakup, droplet collision and coalescence, droplet vaporization, and wall impingement. Consistency between the physical properties of the spray and the chemical model of the fuel is maintained. The advanced techniques employed minimize the dependency on mesh size and time-step size and enables accurate solutions with less computational time compared to conventional spray models.

Different sub-models are applied to represent the spray atomization and droplet breakup processes of solid-cone sprays and hollow-cone sprays, respectively.

For solid-cone sprays, the initial spray conditions at a nozzle exit are specified. These conditions may be either a discharge coefficient or a nozzle-flow model. An unsteady gas jet breakup model is used by default, the Kelvin-Helmholtz / Rayleigh-Taylor (KH/RT) model is used for droplet breakup. Hollow-cone sprays are typically created by pressure-swirl injectors. The injector contains internal swirl vanes that produce rotational motion in the liquid. The liquid forms a film along the inside walls of the injector with an air core in the center. The liquid film becomes a free sheet when exiting the injector, and its tangential velocity becomes radial. As the radial distance from the centerline to the sheet increases, the rotation decreases due to conservation of angular momentum.

Additionally, because of the conservation of mass, the sheet thins as it progresses, and subsequently disintegrates into droplets, forming a hollow cone spray. Once droplets are formed, their behavior is governed by secondary breakup, drag, collision, coalescence and vaporization. Both inward-opening (with an air core but no pintle) and outwardly opening (with a pintle but no air core) are considered.

Dropel collision and coalescence are important physical properties in dense sprays. Which collision outcome occurs depends upon the forces acting on the coalesced pair of droplets. At low Weber numbers, surface forces dominate over liquid inertia forces, and the droplets coalesce permanently. At higher Weber numbers, the liquid inertia forces become more important and the coalescing collisions occur. With further increase of Weber number, the dominant liquid inertia forces cause churning of the colliding droplets, forming a group of small droplets.

All of the outcomes listed above are in ANSYS FORTÉ to provide accurate predictions of the results of a successful coalescence for both solid-cone and...
hollow-cone models. Additional options include a radius-of-influence (ROI) collision model and a collision mesh method. The Radius-of-Influence (ROI) collision model is used to remove both mesh-size dependency and time-step dependency for the droplet collision process. In the ROI method, one particle is allowed to collide with another only if this particle resides within in the radius of influence of the other.

In addition to the ROI model, an alternative approach to mitigating the mesh dependency of the collision model is included. This is the collision mesh model, which employs a pseudo cylindrical collision mesh separate from the CFD computational mesh. ANSYS FORTÉ uses a discrete multi-component (DMC) fuel--vaporization model to represent the vaporization of spray droplets. The DMC vaporization model tracks the individual components (molecules) of an actual surrogate fuel during the evaporation process and allows coupling with the reaction kinetics of the individual fuel components. In the DMC model, an explicit form of the equation that determines the heat flux from the surrounding gas mixture to the droplet--gas interface is obtained from an approximate solution of the quasi-steady energy equation. The model is formulated to track each component of the fuel regardless of the direction of the process, i.e., whether evaporation from the droplet surface or condensation into the droplet is occurring.

**Soot Emissions**

Soot phenomena are notoriously difficult to simulate due to the physics and chemical reactions responsible for soot particle formation. ANSYS FORTÉ predicts soot particle sizes and tracks their progress in an engine simulation. The software’s chemical models follow the reactions from multiple soot precursors through the nucleation, growth, agglomeration and oxidation of particles to predict particle size distributions exiting the engine.

Three methods are provided for predicting soot/particulate properties:

- two-step semi-empirical model, method of moments and pseudo-gas model. The pseudo-gas model include the soot as a pseudo-gas species, and the gas-phase mechanism would contain all the relevant soot nucleation, growth and oxidation pathways.

Although detailed kinetics can be used to predict soot precursors in the gas phase, it is often expedient to use a semi-empirical model to get a qualitative but direct measure of the particle size distribution generated under the simulation conditions of interest. For this reason, ANSYS FORTÉ includes a 2-step semi-empirical soot model with acetylene as the soot precursor that approximates the mass and the particle size of soot formed, based on local species concentrations throughout the simulation.

The Method of Moments model allows tracking of particle size and number density, in addition to soot mass and volume fraction, these variables are determined as a function of location and time through the simulation: it requires more detailed kinetics in the gas-phase fuel combustion model as well as a surface kinetics description of reactions that occur on the particle surface to initiate the formation of dispersed-phase particles. The Method of Moments option requires that a chemistry set be populated with a dispersed-phase particle-reaction reaction, where gas-phase precursors react to form a dispersed particulate bulk phase along with some proportion of occupied and unoccupied surface sites.

**Engine Knock**

Knocking occurs when the highly compressed fuel and air mixture in the combustion chamber auto-ignites, either before or after the spark that is meant to trigger ignition. The spark starts out as an ignition kernel, then quickly transitions into a thin flame surface that moves across the chamber, compressing the unburned gas ahead of the flame. When this unburned fuel--air mixture ignites outside of the planned spark event, it results in knocking. Knock limits engine performance and over time can severely damage the engine. Accurately modeling the location and structure of the flame front as it expands into the combustion chamber, while also allowing for auto-ignition kinetics to occur in the unburned gas, are the main elements in predicting knock. Since the scale of the flame thickness is significantly smaller than the computational mesh, even with very fine grid refinement, CFD simulations that relax on mesh to resolve the flame location will require a large number of tiny cells to resolve the flame topology sufficiently. Meshes with such excessively large numbers of small computational cells required also tiny time steps to maintain simulation stability, which results in an impractical amount of computation time.

ANSYS FORTÉ uses the Discreet Particle Ignition Kernel (DPK) model and the G-equation flame-front tracking technique, to provide fast and accurate resolutions of flame propagation.

When the flame is initiated, the ignition-kernel flame has a structure that is typically smaller than the average grid size in the computational mesh. During this time growth of the ignition kernel is tracked. By assuming a spherical-shaped kernel, the flame front position is marked by Lagrangian particles, and the flame surface density is obtained from the number density of these particles in each computational cell. After the ignition kernel is formed, the calculation switches to the G-equation model. Computational cells with temperatures greater than the critical temperature become ignition sites. When coupled with direct use of detailed chemical kinetics from the Model Fuel Library and ANSYS FORTÉ’s knock-index calculator, the software helps to ensure that engine designers have accurate simulations for predicting auto-ignition and knock propensity in the engine under a range of operating conditions.

**MAGMA 5.3 Fully Optimize Beyond the virtual simulation**

The castings are getting more and more complex in shape, with high performance requirements, and at the same time the robustness of process and high quality. In other words, the casting design requires a compromise of acceptable cost, weight reduction, appropriated mechanical properties, structural integrity and stable process. To achieve these objectives, the Design Chain approach is increasingly applied to evaluate the best solutions in terms of castability and mechanical behaviour. One of the most relevant and Virtual successful innovations in recent years is the Automatic Process Optimization to satisfy all the casting requirements. The new MAGMAS Optimization module is evolving to support the design during the research of the best engineering solution. Automatic optimization aims at calculating and identifying optimized casting process variants. For this purpose, the software automatically varies the rigging system within the bounds of the predefined degrees of freedom. Thus, in contrast to virtual designs of experiments, most experiments are yet unknown at the beginning of the optimization run. On the basis of given objective functions (and secondary conditions, where applicable), the quality of each calculated variant is then evaluated. The automatic variation of the variables defined is initialized based on a given Design of Experiment sequence (start sequence). Depending on the achievement level of the objective function for each experiment separately, new variants are created, simulated and evaluated again. Inspired by the genetic processes in biology, this approach comprises several generations and uses mechanisms of inheritance, mutation and selection.

For this reason, with optimization with MAGMAS Rel. 5.3, the user does not need to have a deeper understanding of the optimization method as the software executes the optimization process automatically. The user then has the functionalities of the Assessment Perspective available for the analysis and the detailed assessment of the virtual designs of experiments carried out.

Le attuali condizioni di mercato impongono alle realtà produttive tempi di riapporto compressi, elevate caratteristiche quantitative e prezzi competitivi, in altre parole la progettazione di un pezzo colato impone di trovare un compromesso tra costi accettabili, riduzione di peso, appropriate proprietà meccaniche ed integrità strutturale in un processo stabile. Per raggiungere questi obiettivi, si tende sempre più ad applicare un approccio di “Design Chain”, così da valutare le migliori soluzioni in termini di costabilità e comportamento meccanico.

Una delle innovazioni più rilevanti e di successo degli ultimi anni, è certamente “L’Ottimizzazione di processo automatica”, in grado di soddisfare tutte le exigenze dei pezzi colati. Il nuovo modulo MAGMAS Optimization è stato integrato per supportare la progettazione durante la ricerca della soluzione ingegneristica migliore.

**Figure 4 – Soot predictions with ANSYS Forte compared closely with experimental measurements**

Figure 1 – MAGMA CAD: Variable geometry preparation

Figure 2 – Objectives definition

L’Ottimizzazione automatica punta a calcolare ed identificare le variabili del processo di calotta. Per questo motivo l’software varia in modo automatico le variabili geometriche e di processo scelte entro i limiti posti dai gradi di libertà predeterminati.

Infatti, in contraddizione con “Design Of Experiments” virtuali, la maggior parte degli esperimenti è ancora sconosciuta all’inizio dell’esecuzione dell’Ottimizzazione.

Sulla base delle funzioni obiettivo date (e dalle eventuali condizioni secondarie), viene quindi valutata la qualità di ogni variante calcolata. La variazione automatica delle variabili definite inizia in relazione ad una sequenza di “Design Of Experiment” (sequenza di inizio). A seconda del livello di conseguimento della funzione obiettiva per ogni singolo esperimento, vengono ulteriormente create, simulate e valutate nuove varianti.

Prendendo spunto dai processi genetici della biologia, questo approccio combina diverse generazioni e usa meccanismi di
MAGMA 5.3

La grande innovazione di questa release e l'integrazione del nuovo Ottimizzatore in tutti gli ambienti come il processamento e il post-processing (immagini). Il nuovo post-processing, basato su grafici 3D (Fig. 3), matrici di correlazione (Fig. 4), costituisce una delle più importanti strumenti per rendere decisioni multi-criterio.

La release 5.3 è disponibile per sistemi operativi a 64-bit come Windows (Win7 e Win 8) e Linux (Red Hat Enterprise 6 e 7, Suse Enterprise 12). GIFA, presso il centro fiere di Düsseldorf, costituisce una delle fiere più importanti per presentare MAGMA 5.3, e tutti i clienti sono invitati a visitare lo stand di MAGMA o a richiedere maggiori informazioni ad EnginSoft, distributor per l'Italia.

Ask the expert on MAGMAsoft
Giampiero Scarpa, EnginSoft
g.scarpa@enginsoft.it

What is System Level Thermo-Fluid Analysis?

When the history of fluid simulation is written, a justifiably large proportion of the tome will focus upon the development of solutions based on, or derivatives of, the Navier-Stokes equations. In fact, what are commonly recognised as three-dimensional methods will likely dominate to the exclusion of all else. As with many histories, this wouldn’t be representative.

The reason is simple: there is a type of fluid simulation that adopts a very different approach that has been serving a range of industries very well for over 50 years. It has, and continues to, enable the safe and efficient design of fluid systems ranging widely in both scale, fluid type and engineering application. This type of simulation is frequently termed 1D system CFD or, more appropriately, system-level thermo-fluid analysis.

A World of Systems, Simulation for that World

In the world of fluid dynamics, the generally accepted view of computer simulation is that of a three-dimensional, geometry, immersed in a computational domain (see Figure 1). This domain is discretized, and the thermodynamics of each cell is approximated by the chosen solution method (of which, the Navier-Stokes equations is the most commonly used).

The reason that this is the most widely recognized approach to Computational Fluid Dynamics (CFD) is because it has proven to be an extremely versatile and effective one. However, it is one best suited to isolated geometries with the effects of any surroundings reduced to idealized boundary conditions. Simulation of more realistic systems, for example the effect of slip-streaming an upstream vehicle on a downstream aerodynamics, has been limited by the available computational power.

Taking another example, for engineers concerned with the flow of fluids through pipes, it would be an approach that could supply a great deal of information about the nature of the flow through a conduit, and perhaps even a conduit-bend-conduit system: but what of a more extensive network? Perhaps one that begins with a series of pumps and then goes on via a number of valves, fittings and branches to a number of delivery points several hundreds of kilometers downstream. Not only does this represent a significantly greater computational challenge to the approach described above, the simulation would require a degree of geometrical knowledge of the individual components that may not yet exist. The paradox is that as that knowledge becomes available, the range of useful applications for the resulting simulations diminishes; and pump selection via simulation is of little value if details of pump geometry are required up front to run that very simulation! Finally, the effort required by a user to set up such a simulation and the time taken to calculate a result would be prohibitive for any practical application.

This class of problem is a broad one. Not only does it cover large scale water, oil or gas distribution networks, it also includes engine lubrication circuits, fuel movement and supply on aircraft of all sizes, ballast systems on ships and submarines and ventilation networks of all types to name but a few. It’s almost more difficult to think of any reasonable sized technology that doesn’t require some type of fluid network.

The question is how best to provide the designers and operators of such systems with the advantages of computer simulation, but in a practicable and useful manner.

Building Blocks

Stated simply, the point of a system of conduits or pipes is to take a product (or products) from one or more sources and deliver it (or the resulting mixture) to one or more receiving points. The delivery point may of course be the source in the case of a closed circuit. Thus, the most obvious characteristic of a pipe system is that it places distinct bounds on the direction a fluid may travel. If the bulk flow is in a direction other than...
Buckling analysis in ANSYS

In ANSYS Workbench there are three methods to determine the buckling load of a structure:
- Linear based eigenvalue buckling analysis
- Nonlinear based eigenvalue buckling analysis (new in R16)
- Nonlinear buckling analysis

Linear based eigenvalue analysis

An eigenvalue buckling analysis allows the calculation of the theoretical buckling load of an ideal elastic structure (without any imperfection). In real world applications, geometric imperfections and nonlinear behaviour prevent a structure from achieving its theoretical buckling load (Fig.1). This kind of analysis allows results to be obtained quickly, but linear buckling generally yields un-conservative results: the buckling load calculated with this method is usually superior to the real buckling load.

Nonlinear based eigenvalue analysis

When performing a linear buckling analysis, the static analysis has to be linear and therefore should not contain geometric, material or contact nonlinearities. With the non-linear based approach, already included in ANSYS APDL and available in Workbench starting from R16, it’s possible to take nonlinearities into account even in the pre-buckling phase, which is in the earlier static analysis. What’s different between the linear-based buckling analysis approach and the nonlinear based buckling analysis one? In both cases the aim is to solve an eigenvalue problem, but the nonlinear-based methodology allows to take into account the variation of the stiffness matrix, due to nonlinearities, during the pre-buckling analysis.

Figure 3 - Eigenvalue Buckling analysis block (fig.2).

The equation solving the eigenvalue problem is now:

\[
([K] + \lambda_i [S]) \psi_i = 0
\]

Where:
- \(\lambda_i\): i-th load multiplier (i-th eigenvalue)
- \(\psi_i\): i-th buckling mode.

When a nonlinear based eigenvalue analysis is set, several aspects have to be taken into account:
- In order to activate this computation methodology, at least one nonlinearity (related to material, geometry or contact) has to be defined in the pre-buckling static analysis.
- Besides the loads defined in the static analysis, it’s also necessary to define at least one load in the buckling analysis, so to proceed with the solution. In order to manage this aspect, the option “Keep Pre- Stress Load Pattern” (fig. 4) has been introduced: setting this property to “Yes”, the same load, already defined in the static analysis, will be applied in the buckling analysis; setting this property to “No”, it will be possible to define new loads in the buckling analysis, different from those applied to the static one (in this release the loads definable in the buckling analysis are: Thermal Condition, Nodal Force, Nodal Pressure, Nodal Displacement).
The load multipliers only scale the loads applied in the buckling analysis; the buckling load is determined as follows:

\[ F_{\text{perturbed}} = F_{\text{restart}} + \lambda F_{\text{buckling}} \]

Where:
- \( F_{\text{perturbed}} \) is the load applied in the buckling analysis (perturbed load),
- \( F_{\text{restart}} \) is the load applied in the static analysis (pre-buckling load),
- \( \lambda \) is the load multiplier,
- \( F_{\text{buckling}} = \) load defined by the user in the buckling analysis if "Keep Pre-Stress Load Pattern" = Yes
- \( F_{\text{buckling}} = \) load defined by the user in the buckling analysis if "Keep Pre-Stress Load Pattern" = No

Application case

In order to compare the results obtained from the three different methods presented above, an example can be taken into account where a cylindrical structure undergoes an external uniform pressure. In particular, we can consider a circular cylinder made of 2024-T3 aluminum alloy which is internally stiffened by 5 rings with a 2 section, and which undergoes a uniform external pressure of 0.1 MPa. The cylinder is constrained in 3 nodes, in order to prevent all rigid translations and rotations.

The linear based buckling analysis provides a critical load multiplier equal to 0.37655. Therefore the buckling pressure is equal to:

\[ P_{\text{buckling}} = 0.1 + 0.1 \times 0.37655 = 0.1377 \text{ MPa} \]

It can be noticed that the value of the buckling pressure achieved with the nonlinear based approach is lower than the value obtained with the linear analysis, and therefore much closer to the reference value.

Determination of the buckling pressure with nonlinear static analysis

Using the upgeom operation, a perturbed mesh has been created, which is based on the first ten buckling modes (calculated with the linear buckling analysis), each one of them scaled by a 0.1 factor. The small implemented APDL macro is presented here:

```
$prep7
s_factor=0.1
*upgeom,s_factor,1,i,file, rst ! Add imperfections as a tenth of each mode shape
cdwrite,db,cylinder_perturbed,cdb
*enddo
```

The reference value of the buckling pressure for the specific cylinder is 0.121 MPa: this value comes from an experimental test performed on the real cylinder (ref: Dow, Donaldson A., November, 1965, "Buckling and Post-buckling Tests of Ring-Stiffened Circular Cylinders Loaded by Uniform External Pressure.", NASA Technical Note NASA TN D-3111, Langley Research Center).

The load multiplier only scale the loads applied in the buckling analysis; the buckling load is determined as follows:

\[ F_{\text{perturbed}} = F_{\text{restart}} + \lambda F_{\text{buckling}} \]

where:
- \( F_{\text{perturbed}} \) is the load applied in the buckling analysis (perturbed load),
- \( F_{\text{restart}} \) is the load applied in the static analysis (pre-buckling load),
- \( \lambda \) is the load multiplier,
- \( F_{\text{buckling}} = \) load defined by the user in the buckling analysis if "Keep Pre-Stress Load Pattern" = Yes
- \( F_{\text{buckling}} = \) load defined by the user in the buckling analysis if "Keep Pre-Stress Load Pattern" = No

Determination of the buckling pressure with nonlinear based eigenvalue analysis

In order to apply the nonlinear based computation methodology, geometric nonlinearities have been activated and a material plastic behavior (bilinear isotropic hardening) has been defined. The buckling, nonlinear based analysis provides a critical load multiplier equal to 0.37655. Therefore the buckling pressure is equal to:

\[ P_{\text{buckling}} = 0.1 + 0.1 \times 0.37655 = 0.1377 \text{ MPa} \]

The “Keep Pre-Stress Load Pattern” option was set to YES, therefore the applied pressure in the buckling analysis is the same as the one applied in the static analysis (0.1 MPa).

The following table presents a synthesis of the results that have been achieved using the three methods (Table 1). It can be noticed that the most accurate result is the one provided by the nonlinear analysis. Nevertheless the new Nonlinear based eigenvalue analysis allows to achieve a result much closer to reality in comparison with the linear method, because it can account for the nonlinearities in the pre-buckling phase.

<table>
<thead>
<tr>
<th>Method</th>
<th>Buckling pressure [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear based eigenvalue analysis</td>
<td>0.1422</td>
</tr>
<tr>
<td>Nonlinear based eigenvalue analysis</td>
<td>0.1377</td>
</tr>
<tr>
<td>Nonlinear buckling analysis</td>
<td>0.1218</td>
</tr>
<tr>
<td>Reference value</td>
<td>0.121</td>
</tr>
</tbody>
</table>

Conclusions

The following table presents a synthesis of the results that have been achieved using the three methods (Table 1). It can be noticed that the most accurate result is the one provided by the nonlinear analysis. Nevertheless the new Nonlinear based eigenvalue analysis allows to achieve a result much closer to reality in comparison with the linear method, because it can account for the nonlinearities in the pre-buckling phase.
Growing food for space exploration:
EnginSoft is now part of the EDEN ISS project

A new project is underway under the European Union's Research and Innovation Action program Horizon 2020, within the topic of 'Space exploration/Life support.' Thirteen international organizations, including universities, corporations and small businesses, in the cross-disciplinary fields of food growth, life support, engineering and design come together in taking one step further towards the independent exploration of celestial bodies such as the Moon or Mars. During the four year collaborative project, the group will be developing new closed-loop systems for food cultivation.

EDEN ISS is a project focused on the 'Ground Demonstration of Plant Cultivation Technologies and Operation in Space' and the enhancement of those technologies, 'For Safe Food Production onboard the International Space Station (ISS) and Future Human Space Exploration Vehicles and Planetary Outposts.' The consortium is led by the German Aerospace Center (DLR) Institute of Space Systems in Bremen, Germany and includes the following partners:

- Telespazio S.p.A., Italy
- Limerick Institute of Technology, Ireland
- Heliospectra AB, Sweden
- Wageningen University and Research, the Netherlands
- Aero Sekur S.p.A., Italy
- Thales Alenia Space Italia S.p.A., Italy
- Airbus Defence and Space, Germany
- EnginSoft S.p.A., Italy
- Alfred Wegener Institute for Polar and Marine Research, Germany
- University of Guelph, Canada
- National Research Council, Italy
- LIQUIFER Systems Group, Austria
- DLR Institute of Aerospace Medicine in Cologne, Germany

Collaborative project, the group will be developing new closed-loop systems for food cultivation.

The overall goal of the EDEN ISS project is to advance the key technologies in controlled environment agriculture in the following areas; an advanced nutrient delivery system, a high performance LED lighting system, a bio-detection and decontamination system for food quality and safety.

A mobile, container-sized, greenhouse test facility will be built to demonstrate and validate different key technologies and procedures necessary for safe food production within a (semi-) closed system.

The service section will house the main support subsystems, including thermal, power, air ventilation and nutrient/water subsystems and will provide working space for pre and post-harvest procedures. The International Standard Payload Rack (ISPR) section will contain two rack systems for plant production (similar to the rack-type used on the ISS) and the Future Exploration Greenhouse (FEQ) will consist of a highly adaptable multi-shelf growth system, capable of maintaining a number of different environmental settings.

The plant cultivation technologies will first be tested in a laboratory setting at the sites of the consortium partners. All systems will be integrated at DLR in Bremen, followed by an extensive test period. In October 2017, the complete facility will be shipped to the German Neumayer III station in Antarctica. The station is operated by the Alfred-Wegener-Institute and has unique capabilities and infrastructure for testing plant cultivation under extreme environmental and logistical conditions. It is foreseen that the container-sized greenhouses of the EDEN ISS project will provide year-round fresh food supplementation for the Neumayer Station III crew.

For more information: Erik Mazzoleni, EnginSoft e.mazzoleni@enginsoft.it

Bringing Educational Robotics in schools
Introducing robotics activities in the existing didactical curriculum

For the third year, the IAS-Lab (Intelligent Autonomous Systems Laboratory) of the University of Padua (Italy) is running an intensive training course for school teachers about the use of Educational Robotics. The course will run for 3 full-days in September 2.3 and 4, 2015. Educational robotics (ER), though not widely known and adopted in the Italian school system, is no more a sort of exotic technology. Robotics offers a new way of teaching to students, especially when integrating it into standard class curricula. Experiences in scientific literature and several pilot projects show its relevance as a valuable multidisciplinary tool for any age and any kind of school, from kindergartens to university and beyond. Educational Robotics strongly motivates project-based learning and team working in open teaching/learning scenarios, whatever specific activities are designed, from simple laboratory experiences to structured robotic competitions. Educational Robotics can be sure foster the interest for scientific and technical disciplines, but its impact is not limited to science, technology, engineering, and mathematics (STEM). It proved to be effective, motivating students and young pupils in the learning of geography, foreign languages, road-safety education and philosophy.

The course headed by Prof. Michele Moro with the collaboration of prof. Emanuele Menegatti will cover the methodological and foundational aspects for the educational use of robotics in schools. It will also provide a large experience in the lab with real robots. The course is aimed at teachers of schools of all levels, usually equally divided among primary, middle, and secondary schools.

The teachers will be guided to get acquainted with the technological aspects of robot programming, and then will practice in the lab simulating activities with their students. Starting from typical didactical needs in schools, illustrative examples of lab activities are explained and the teachers will practice implementing them. In the end, teachers, in small groups, will design their own robotics activity based on the needs of their students and this proposal will be presented and discussed with the other participants. Some private companies recognizing the importance of promoting the use of robotics in schools are actively supporting this course aimed at the professional development of teachers.

This support is realized by covering the registration fees of teachers whose schools do not have funds to directly register for the course. This year EnginSoft is proudly sponsoring the course by covering several teacher registration.

For more information: http://thekylor.com/roboticaeduttica2015

Figure 1 - EDEN ISS team photo, Credit: DLR 2015

Figure 2 - Greenhouse test facility, Credit: LIQUIFER Systems Group 2015
At the International CAE Conference, EnginSoft, for the fourth year in a row, champions and sponsors the “Poster Award”, a contest dedicated to recognizing the use of CAE technologies that are illustrated on posters.

The purpose is twofold: offering recognition to excellence and innovation of projects developed in the academic field, and providing a privileged meeting space between the university experiences and the industrial world.

www.caeconference.com

EnginSoft, as coordinator of the MUSIC consortium and developer of software and database, was invited to participate in the CO.SUMMIT 2015, to present for the first time a prototype of the MUSIC Control & Cognitive System. ICT, Communication and software innovation for next generation of Factory of Future are the key elements of MUSIC project.

The Control & Cognitive (C&C) system will activate a cost efficiency loop in the high pressure die casting of light alloys (HPDC) and plastic injection molding (PIM) industry by introducing for the very first time an holistic approach to real time data monitoring, analysis and control of all the phases of the currently fragmented automated production lines.

MUSIC Project at CO.SUMMIT 2015
last 10/11 March 2015 in Berlin
The 7th edition of the Co-summit was dedicated to ‘Smart industry: impact of software innovation’

MUSIC Tour at GIFA
The MUSIC project invites the visitors of GIFA International Foundry Trade Fair to visit its partners’ booths to get acquainted with the research activities and the last achievements in the High Pressure Die Casting and Plastic Injection Moulding sectors.

For further information the MUSIC project: http://music.eucoord.com

MUSIC Control & Cognitive System
Functional Specifications

MUSIC Tour at GIFA
The MUSIC project invites the visitors of GIFA International Foundry Trade Fair to visit its partners’ booths to get acquainted with the research activities and the last achievements in the High Pressure Die Casting and Plastic Injection Moulding sectors.

For further information about MUSIC project dissemination activities and to download the flyer of the event, please go to: http://music.eucoord.com/calendar/body.pe

For further information the MUSIC project: http://music.eucoord.com
Save the date

19 - 20
October 2015
Pacengo del Garda,
Verona - Italy

Your opportunity to be part of the future!

The annual Simulation Based technology event, will once again present technically rich content to the satisfaction of our esteemed guests.

This year’s plenary session will honour EXPO Milano, with stimulating speeches relating to the food chain industry, such as international projects referring to astronaut food product.

Plenary sessions will feature the futuristic building designed for the EXPO, which was verified using sophisticated FEM calculations. These sessions will be followed by fascinating topics delivered by international key note speakers on: numerical analysis applied to artwork maintenance, exciting innovative project related to the exploitation of renewable energy.

The plenary will be followed by several transversal sessions, focusing on the wide range of simulation sectors such as: Transportation, Energy and Oil & Gas, Aerospace & Defence, Construction, Manufacturing, Foundry and Metal Forming, Explicit Dynamics, Healthcare, ITC & HPC.

Among this year’s edition, attendees will have access to software experts in the Technology Gallery, where there are opportunities to book onsite technical demonstrations and to meet major hardware and software developers in the exhibition area.

For all updates, stay tuned!

www.caeconference.com