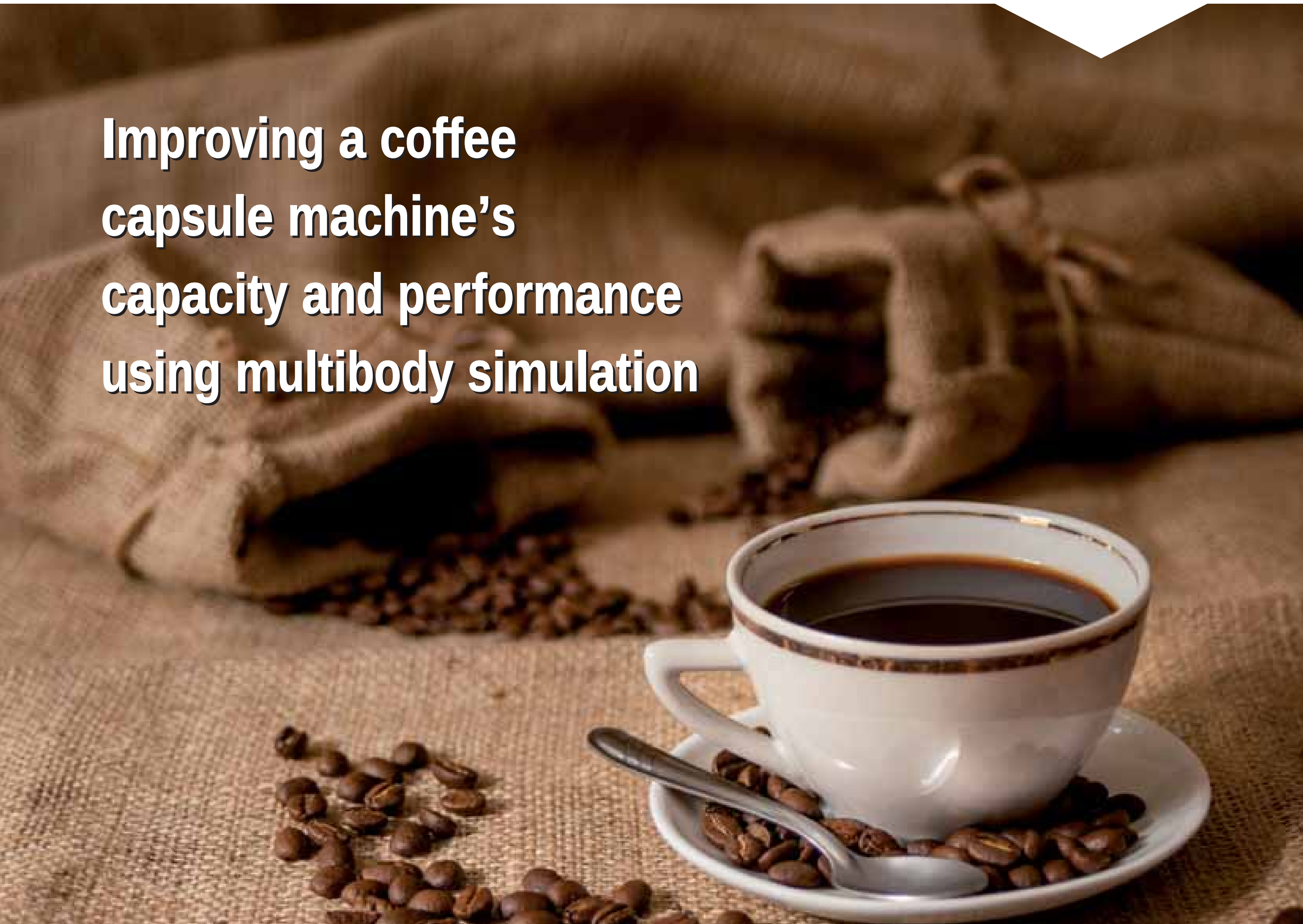
ENGINSOFT

The Simulation Based Engineering & Sciences Magazine

Newsletter

Year 16 n°3

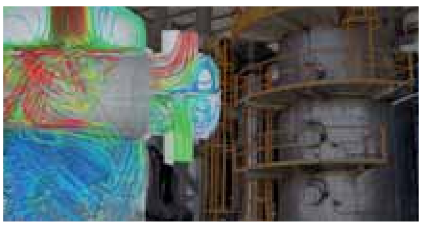
Autumn 2019



Improving a coffee capsule machine's capacity and performance using multibody simulation

Refining the design of a spout for a laundry detergent

The future of vehicle systems engineering



Optimizing gas flow for the transformation process of chemicals

apave italia cpm
www.apave.it - info.it@apave.com

(•) Sede Legale e Operativa
25040 BIENNO (BS)
Via Artigiani, 63
Tel. +39.0364.300.342
Fax +39.0364.300.354

(▲) Ufficio di Arcore
20862 ARCORE (MB)
Via Forlanini, 52
Tel. +39.039.389.696
Fax +39.039.389.947

Ufficio di Roma
00155 ROMA
Via B. Bardanzellu, 94
Tel. +39.06.332.70123
Fax +39.06.332.20293

Ufficio di Padova
35133 PADOVA (PD)
Via T. Aspetti, 170 int. 9
Tel. +39.049.605.446
Fax +39.049.889.9164

Con più di 250 ispettori noi offriamo ai nostri clienti i seguenti servizi:

ORGANISMO NOTIFICATO EUROPEO

- Recipienti semplici a pressione •
- Attrezzature a pressione (PED) •
- Attrezzature a pressione trasportabili T-PED •
- Giocattoli •
- Macchine ▲
- Ascensori •
- Prodotti da costruzione: *aggregati, miscele bituminose, prodotti metallici, porte e cancelli*

CE 0398

VERIFICHE PERIODICHE REGOLAMENTARI

- Messa a terra •
- Ascensori •
- Pressione
- Sollevamento

LABORATORIO AUTORIZZATO LEGGE 1086/71

Prove su materiali per strutture metalliche e per le opere in cemento armato

ORGANISMO DI CERTIFICAZIONE DM 17.01.2018

Certificazione impianti di calcestruzzo industrializzato

LABORATORIO PROVE E MISURE

Meccanico, edile, chimico, ambiente, elettrico

LABORATORIO PROVE MECCANICHE

• ACCREDIA LAB. N° 1400

UNI CEI EN ISO/IEC 17025

Accreditamento limitato alle prove elencate sul sito www.accredia.it nella pagina dedicata al laboratorio APAVE ITALIA CPM. L'accreditamento del laboratorio APAVE ITALIA CPM si riferisce alle prove e non ai prodotti oggetto di esame.

SANIFICAZIONE

Certificato UNI EN ISO 9001 - Settore EA35

•▲ Servizi accreditati **ACCREDIA**

ENTE DI VERIFICA INDIPENDENTE

- Controllo tecnico costruzioni edili
- Elettricità e radiazioni
- Energia e fluidi
- Salute e sicurezza sul luogo di lavoro
- Macchine ed impianti
- Analisi ambientali
- Gestione dei rischi
- Sicurezza alimentare

ENTE DI FORMAZIONE

Certificato UNI EN ISO 9001 - Settore EA37

- Organismo accreditato per i servizi di istruzione e formazione professionale Regione Lombardia - Iscrizione n. 527 del 02/03/2009 - Sezione B
- Organismo accreditato per i servizi al lavoro Regione Lombardia - Iscrizione n. 160 del 07/07/2009 *ai sensi del D.D.U.O. n. 9749 del 31/10/2012*

ENTE DI CERTIFICAZIONE

• ACCREDIA PRS N° 079 C

Certificazione delle competenze UNI CEI EN ISO/IEC 17024

- Saldatori, operatori di saldatura, addetti alle prove non distruttive
- Personale che interviene sui gas-fluorurati
- Personale che interviene su impianti GAS - UNI 11554
- Personale che opera su impianti di refrigerazione e pompe di calore - UNI 13313
- Esperto in Gestione dell'Energia (EGE) - UNI 11339
- Tecnici per la ricostruzione e l'analisi degli incidenti stradali - UNI 11294

ACCREDIA L'ENTE ITALIANO DI ACCREDITAMENTO

PRD N° 130 B
ISP N° 093 E
PRS N° 079 C
LAB N° 1400

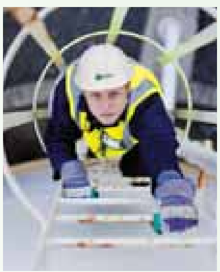
Membro degli Accordi di Mutuo Riconoscimento EA, IAF e ILAC

Signatory of EA, IAF e ILAC Mutual Recognition Agreements

Groupe apave

Società mondiale di ispezione con 10.860 dipendenti che dal 1867 impegna le proprie strutture al controllo di impianti e prodotti con un'offerta globale nella gestione dei rischi.

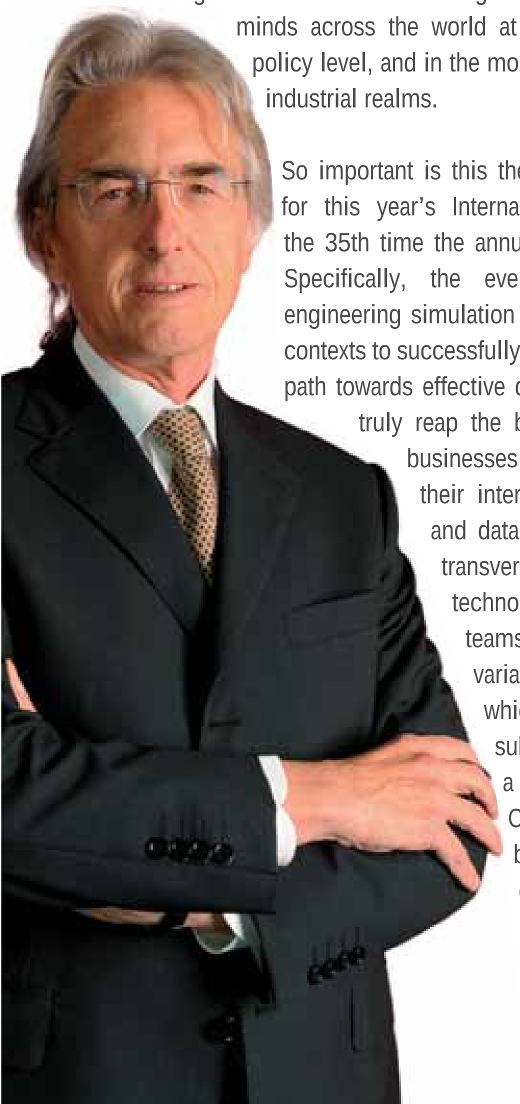
www.apave-international.com




Flash

Gartner's glossary defines digitalization as "the use of digital technologies to change a business model and provide new revenue and value-producing opportunities. It is the process of moving to a digital business."

This is increasingly becoming important for businesses and government's alike, for improved customer service, greater product, production and building efficiency, cost reduction, and reduced time-to-market. For businesses, possibly its greatest allure lies in the promise of the facilitation of innovation – as more data is available about products from their design, through their production, up to their end-of-life, and as that can be correlated with the data from customers concerning how they select, use and maintain those products, the hope is that artificial intelligence, deep and machine learning will enable business strategists and engineers to identify sweet spots for innovation. This is the utopian vision. Plotting a path for your business to become digital and obtaining these benefits are challenges that are occupying many minds across the world at political and economic policy level, and in the more practical business and industrial realms.



So important is this theme, that it is the focus for this year's International CAE Conference, the 35th time the annual event will take place. Specifically, the event will consider how engineering simulation can be used in different contexts to successfully orchestrate a considered path towards effective digital transformation. To truly reap the benefits of digitalization, businesses need to break down their internal silos of information and data and create a culture of transversal integration across technologies, products, and teams. There are many variables to consider, each of which represents an entire subject of study. Attending a leading event like the CAE Conference gives business and technology decision makers a chance to review the newest technologies on the market and learn more about the ones that best fit your business imperatives. More importantly, however, it brings together thought and business and engineering leaders from around the world, offering the opportunity to discuss approaches, learn from colleagues, network and compare practices and experiences.

The organizers have substantially increased the value of this year's event by launching an independent industry and technology trade show alongside the conference, which has attracted the participation of a wide variety of vendors and service providers involved in the digitalization of industrial processes for industry 4.0, as well as from companies that interconnect with these technologies in some way. The exhibition has been organized to enable the exhibitors to offer their own mini-events, workshops and presentations to participants, directly from their stands.

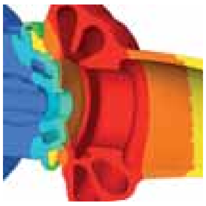
The conference program itself is very broad and relevant, including a rich plenary session and presentations from the energy, oil and gas, automotive, aerospace and defense and manufacturing sectors. The range and relevance of the many independently organized collateral events in the program also attest to the important turning point that the International CAE Conference and Exhibition has reached this year. The collateral events cover a broad spectrum of timely topics. Some of these include a civil engineering event endorsed by the CNI (the National Council of Italian Engineers), an event organized by APRE (Agency for the Promotion of European Research) on developing innovation through the Horizon 2020 projects, the patronage of SIMAI (the Italian Society for Applied and Industrial Mathematics), and the very prestigious round table event featuring some of Italy's most successful female engineers – Maria Antonietta Perino, Monica Valli, Chiara Montanari, and Alessandra Angelini – which will discuss how to increase the participation of women in impactful roles across the engineering sectors.

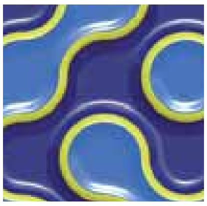
With such a rich program, as well as all the structured networking opportunities, such as the workshops and round table events, in addition to the informal networking events, this year's CAE Conference and Exhibition is a must-attend event. EnginSoft will have a strong presence. I look forward to seeing you there

Stefano Odorizzi,
Editor in chief

Contents

Case Studies





Interviews

Software Updates

Press Release

Newsletter EnginSoft
Year 16 n°3 - Autumn 2019

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

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

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

The EnginSoft Newsletter is a quarterly magazine published by EnginSoft SpA

COMPANY INTERESTS
EnginSoft GmbH - Germany
EnginSoft UK - United Kingdom
EnginSoft France - France
EnginSoft Nordic - Sweden
EnginSoft Turkey - Turkey
VSA-TTC3 - Germany
www.enginsoft.com

CONSORZIO TCN www.consorziotcn.it • www.improve.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it
AnteMotion

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

ADVERTISING
For advertising opportunities, please contact our Marketing office at: info@enginsoft.com

RESPONSIBLE DIRECTOR
Stefano Odorizzi

PRINTING
Grafiche Dalpiaz - Trento

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

Improving a coffee capsule machine's capacity and performance using multibody simulation

Improvements achieved by optimizing laws of motion governing machine components

by Davide Marini
EnginSoft

Capacity is the most important feature of an automatic packaging machine since it defines its main competitive advantage. OPEM's designers are constantly looking for new solutions to increase the capacity of their machines without compromising costs, size and efficiency.
This project focused on a machine for the production of coffee capsules. The main objective was to increase capacity by 25% without affecting the architecture and size of the machine. The machine's operations can be divided in two main areas:

1) Stationary operations
2) Moving operations

The filling and sealing of capsules are "stationary operations". During these operations, the capsule must remain stationary for a prescribed amount of time. The forward movement of the capsules, and the feeding, cutting and transport of the sealing film are "moving operations". Stationary operations were not addressed by this project (i.e. the time constraint resulting from these operations remained the same). As a result, the cycle time for performing the moving operations was reduced by well over 25%.

To achieve the desired objective OPEM partnered with EnginSoft to optimize the laws of motion governing different mechanisms. The optimization process was supported by multibody simulations performed in RecurDyn to verify the dynamic behavior of systems subject to these new and highly demanding laws of motion.

Units analyzed

The mechanisms analyzed can be divided into four units:

- Cut and weld
- Film feeder
- Conveyor
- Structure

The cut and weld unit (Fig. 1) cuts the film into circular portions and transports and seals them on top of the capsules. To perform these operations, the components of the unit are moved by two electric motors. The motion applied by the electric motors is transferred through a series of four-bar linkage mechanisms and crank mechanisms, which determine a nonlinear relationship between the movement of the motor and the motion of the end effector.

The film feeder (Fig. 2) supplies the film to be cut and sealed. The movement of the film is controlled by several electric motors to apply the correct tension and to align the film correctly. When performing the cutting operation, the nesting of the holes is very important to reduce the amount of scrap material. OPEM's designers included a crank mechanism which translates the entire film feeder unit resulting in better nesting of the holes and thus reducing scrap material (Fig. 3).
The current study focused on this lateral movement of the unit since the advancement of the film is already well-known to the OPEM designers and therefore no further study was requested on this topic.

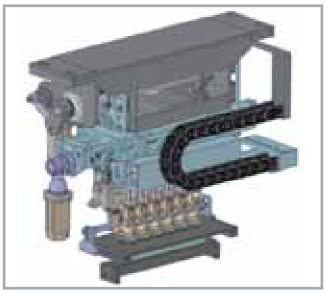


Fig. 1 - Cut and weld unit

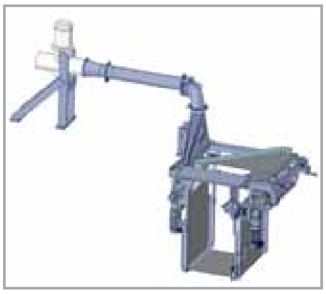


Fig. 2 - Film feeder

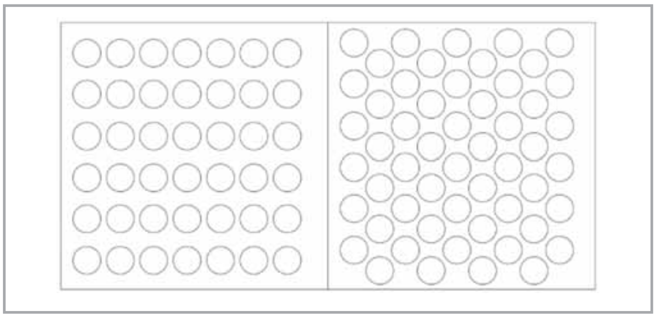


Fig. 3 - Examples of nesting. The solution on the right, obtained by laterally translating the film, clearly reduces the scrap material.

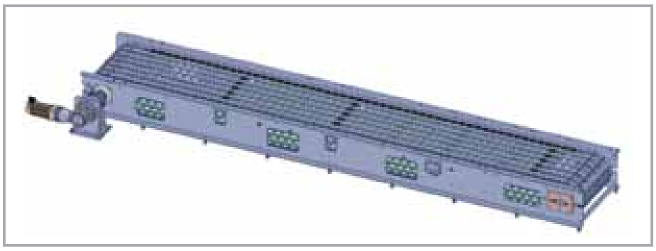


Fig. 4 - Conveyor

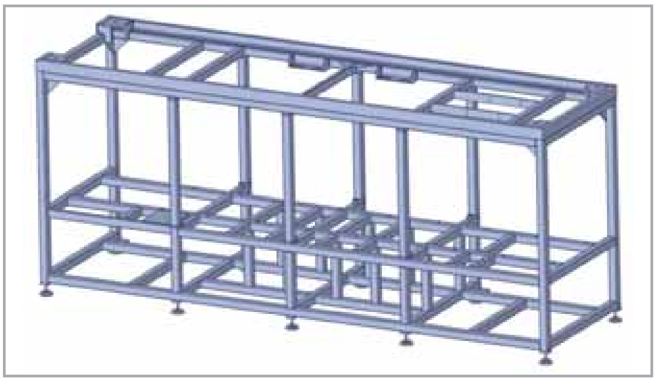


Fig. 5 - Structure

The conveyor (Fig. 4) moves the capsules. The empty capsules are automatically loaded at one end of the machine. At the beginning of each cycle the capsules are moved one step forward until they reach the other end of the machine where they are unloaded. The capsule supports are fixed to two chain mechanisms consisting of several hundred chain links, supporting guides and four sprockets. An electric motor is directly connected to two of the four sprockets and rotates them at the beginning of each cycle to advance the chain mechanism.
The last unit analyzed was the structure (Fig. 5). Built mainly of beams, it supports all the other units.

Optimization of the laws of motion

The core activity of this project was to optimize and validate the laws of motion governing the cut and weld unit, the film feeder and the conveyor. This was done using an iterative approach:

1) EnginSoft internally developed routines were used to generate an initial set of laws of motion
2) The laws of motion generated were applied in the RecurDyn models to control the virtual motors and the unit simulations were performed under dynamic conditions
3) Engines torque, position, speed and the acceleration of the end effectors were measured

The laws of motion were optimized repeating points 1÷3 with the objective of minimizing both the motors torque and the end effectors' acceleration. (At the same time, the desired position of the end effectors were applied as a constraint and the angular velocities of the electric motors were monitored and compared to the prescribed threshold levels).

The optimization was simple for the conveyor unit because the electric motor is directly connected to the sprockets that control the position of the capsules. The acceleration of the motor and the end effector (i.e. the capsules) can be minimized at the same time by choosing a law of motion that minimizes the acceleration of one of the two (the relationship between the motor and the end effector is linear and equal to the transmission ratio since the chain dynamics and the transmission error are negligible in this application). A modified trapezoidal motion law was used to control the electric motor because this profile guarantees the minimum acceleration value (Fig. 6) compared to other motion laws (cycloidal, polynomial and others).

The optimization process was more complex for the cut and weld unit and the film feeder because the four-bar linkage and the crank mechanisms create a nonlinear relationship between the motors and the end effectors. Consequently, it is not possible to minimize the acceleration of the motors and the end effectors at the same time by studying only one of them. Under these conditions, the iterative approach reported at the beginning of this paragraph was crucial to optimize the motion law of both the end effectors and the electric motors.
The optimal law of motion was obtained by using a generalized trapezoidal profile and by iteratively tuning the length of the acceleration / constant speed / deceleration intervals. Fig. 7 shows the displacement, speed and acceleration of the cut and weld end effector: black lines are obtained when the acceleration of the end effector is minimized, red lines are obtained when the acceleration of the motor is minimized and green lines are the result of the optimization process.

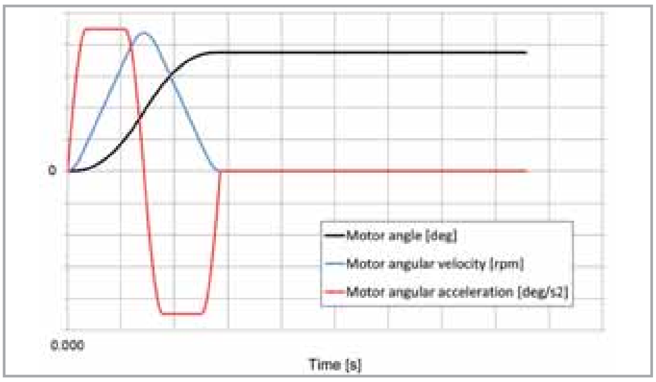


Fig. 6 - Modified trapezoidal motion law

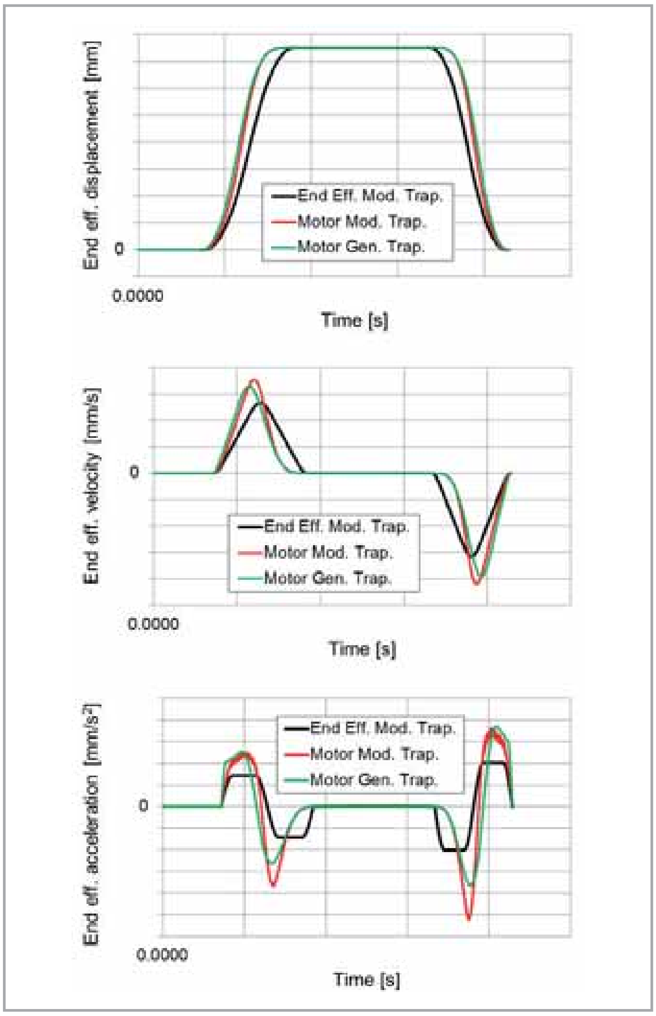


Fig. 7 - Position, speed and acceleration of the cut and weld unit end effector.

Modelling

The optimization process of the laws of motion described in the previous paragraph was assisted by the results of the multibody models developed in RecurDyn.
The cut and weld unit was initially represented using a multibody model made of rigid bodies only. This model was suitable for use during the optimization process because the time required for the calculation was less than 20 seconds. This model was then used to optimize the laws of motion for the two electric motors, and also to verify their synchronization. (The synchronization of the motors is very important to perform a precise cut of the film and to avoid collision with the other components). Once the optimization process was complete, several components of the cut and weld unit were transformed from rigid to flexible bodies (Fig. 8). The simulation was repeated to calculate the stress acting on the components under dynamic conditions. No structural problems were found demonstrating that the optimization of the laws of motion was able to increase the performance of the machine as required, minimizing the variation of internal loads. At the same time, this result proved the robustness of the machines designed by OPEM.

The model for the film feeder was not developed starting from rigid bodies alone: the flexibility of the film was taken into account from the outset to ensure that the modification of the law of motion did not affect

Fig. 8 - Overview of the RecurDyn model of the cut and weld unit including flexible bodies

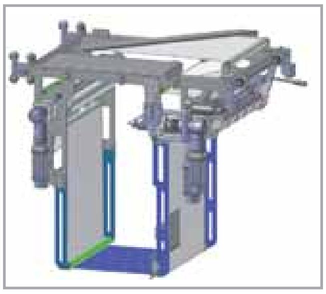


Fig. 9 - Overview of the RecurDyn model of the film feeder including flexible bodies

its dynamics and final position. The film was modelled using RecurDyn's proprietary Full Flex approach to correctly represent the film's large displacements during lateral movement. (Full Flex technology transfers the full stiffness matrix of an object to the RecurDyn solver, i.e. all the degrees of freedom (d.o.f) of the flexible bodies are taken into account during the solution, which allows the correct representation of nonlinearities such as: large deflections, nonlinear materials and contacts over flexible surfaces). Once the optimization process was completed, the side plates of the unit were transformed from rigid to flexible (Fig. 9) to calculate both the stress acting on these components, and to assess whether their flexibility could cause unwanted lateral oscillations of the film at the end of the movement.

Once again, the optimization process led to a law of motion able to meet the requirements without introducing structural problems. In addition, the interaction between EnginSoft and OPEM's engineers highlighted some areas where the weight of the unit could be further reduced to achieve even better performance in the future.

The multibody model developed to study the conveyor included more than 300 contact points since all the chain links are supported without the use of joints. Their interaction with the sprockets and guides relies only on the contacts. This model was used to calculate the motor torque, to simulate the chain dynamics and to calculate the chain tension resulting from the preload and the conveyor's movement.

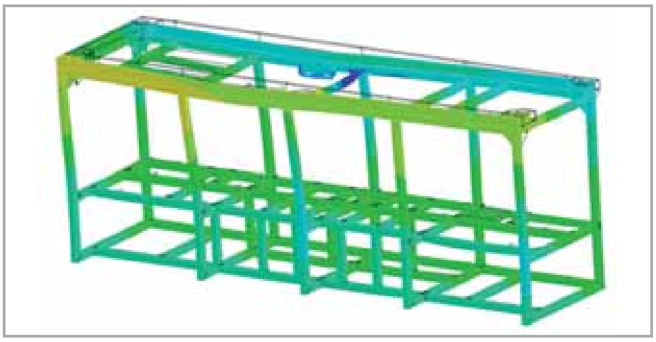


Fig. 10 - Magnified view of the frame's deformation

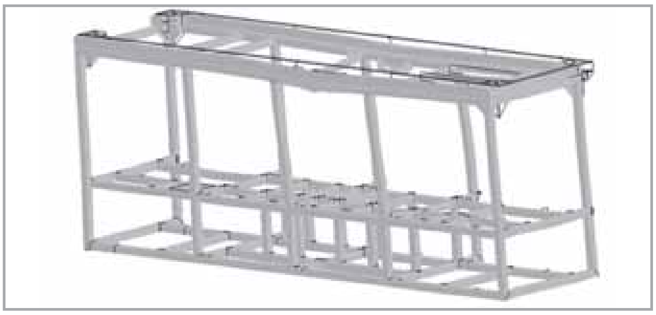


Fig. 11 - von-Mises stress contour

The loads obtained from the multibody models were subsequently used for the structural verification of the frame. A finite element model was constructed using ANSYS Workbench to calculate both the deformation and the stress acting on the structure (Fig. 10 and Fig. 11).

Conclusion

This study successfully improved the capacity of a capsule machine by 25% by optimizing the laws of motion governing the mechanisms that compose it.

The optimization process was based on an iterative approach supported by multibody models developed in RecurDyn. These models were also used to correctly size the main components of the machine, to verify its dynamic behavior when subjected to the new laws of motion and to obtain the loads acting on the frame, which were subsequently used for structural verifications.

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

About OPEM

OPEM S.P.A was founded by Fabio Binacchi in 1974 in Parma as a small, family-run company which started with the purchase of a pasta business that laid the foundations of the future OPEM. In the beginning, the company had ten employees and the coffee market was still on the distant horizon, but continuing requests for new weighing and packaging machines led the company to study and design systems that could be used in different sectors. Binacchi started working for third parties and became interested in packaging not only for pasta and biscuits (which were never completely abandoned), but also for coffee. In the Eighties, OPEM worked with important customers such as Barilla and Kimbo and, over the years, it expanded its horizons abroad, working with many international customers. The rise of the rapidly booming market for pods and capsules gave the company the opportunity to grow and expand, thanks to their foresight in producing very flexible systems that can be customized to meet many customers' requests. In 1984, the company moved to Parma's industrial area and employed 105 workers. Thirty years later, after more than three years of construction, the new OPEM plant was inaugurated in Parma's industrial area in September 2014, proudly affirming the confidence in the company's know-how and its commitment to the future.

Using multi-disciplinary optimization for engine suspension stiffness

Optimizing idling and ride comfort

By Stefano Pizzuto
Fiat Chrysler Automobiles

In this technical article, Fiat Chrysler Automobiles explain how they created a multibody optimization project to identify the optimal values for the powertrain suspension stiffness for a three-cylinder engine in order to minimize the vibrations at idle condition and ensuring greater ride comfort to the passengers.

A vehicle's engine suspension is primarily responsible for shielding the body of the car from vibrations generated in the engine by the reciprocating masses and the forces of the gas pressure. For vibrations during engine idling, this is mainly achieved by tuning the stiffness of the engine mounts, so that the natural frequencies of the system are set far enough away from any frequencies that may be produced by the engine. During operation of the car, however, the excitation induced by movement on the road produces undesirable shaking in the engine, that may cause perceptions of discomfort to passengers. The tuning requirements for these two cases (the idling engine and the shaking engine in motion) can often generate opposing choices. To harmonize these choices, we conducted a multidisciplinary optimization on the stiffness of the engine mounts using modeFRONTIER® and ADAMS/Car®, for a three-cylinders engine. A preliminary design of experiments (DoE) was used to train a set of response surfaces, after which a genetic algorithm was used to find the Pareto frontier (response surfaces method - RSM).

Problem description

Internal combustion engines generate vibrations during operation. The first class of excitation is due to the design parameters: inertial forces and torques from reciprocating masses, variations in gas pressure in the cylinders, centrifugal forces. A second class of excitation is generated by variations in the production process: combustion problems, crank mechanism unbalanced components, unbalanced ancillaries.

A powertrain suspension is used to shield the car body from these vibrations.

Fig.1 shows a typical engine suspension for a front-mounted East/West engine: two mountings in the vertical plan of the center of gravity and a torque rod.

In the case of a three-cylinder engine without countershafts, the engine excitation (Fig.2) can be described as follows:

- Order 0.5: Combustion and cylinder-to-cylinder irregularities
- Order 1: Unbalanced yaw irregularities
- Order 1.5: Combustion (3 burns during 2 revolutions)

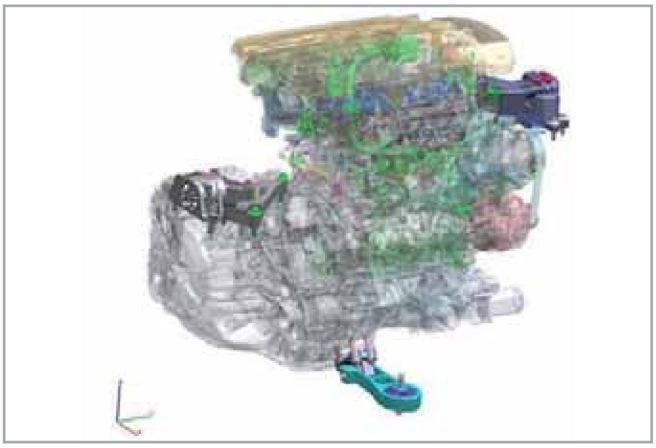


Fig. 1 - 2 mounts – 1 rod powertrain suspension

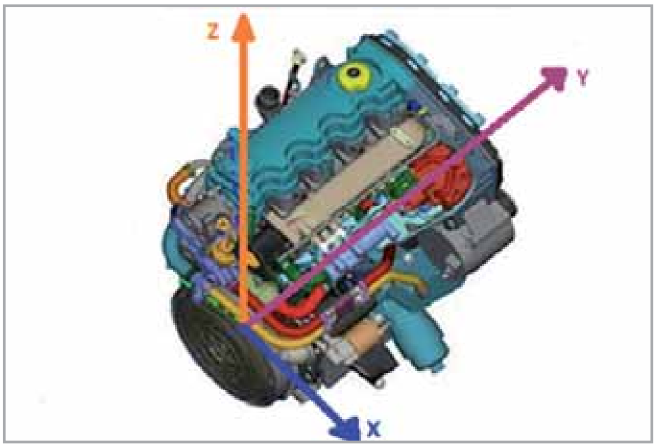


Fig. 2 - Reference system

During a car journey, road irregularities are transmitted to the powertrain, which has at least 6 degrees of freedom as a rigid body within its bay – due to the engine's suspension which allows it to move under both static and dynamic loads. The energy of these engine vibrations is transmitted back to the car body, which can cause vibrational discomfort for passengers. This is called "engine shake".

Fig.3 provides an example of a modal analysis obtained by linearizing the system at amplitudes of ride comfort, while

n.	frequency [Hz]	direction	description
1	6.5	Rx+Y	roll + transverse
2	6.8	Z+X	vertical shake + longitudinal
3	8.1	X+Z	longitudinal + vertical shake
4	11.4	Rx+Rz	roll + yaw
5	12.2	Z+Y	vertical shake + transverse
6	14.6	Rz+Ry	yaw + pitch

Fig. 3 - Example of one modal analysis of engine shake

Fig.4 shows the power spectral densities (PSD) of the vertical accelerations on the wheel hub, car body and engine during a ride comfort mission.

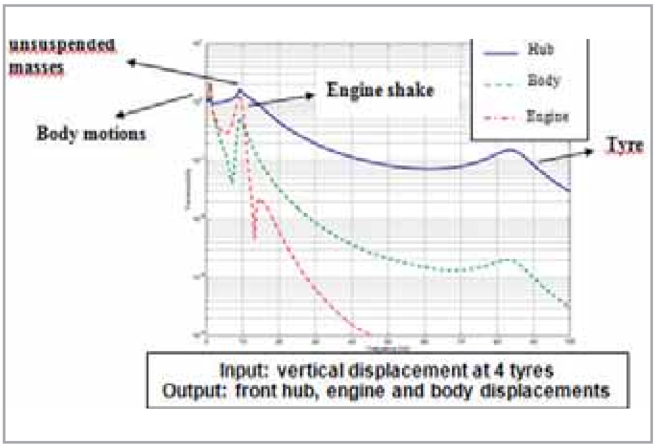


Fig. 4 - PSDs of a ride comfort mission and engine shake

Description of the virtual model

The models used for this study are multibody models of complete vehicles, created in MB-Sharc®, a customized version of MSC Adams/Car® software developed by Centro Ricerche Fiat. MB-Sharc® uses Fiat Chrysler Automobiles (FCA) content for the suspension templates, components, maneuvers and post-processing.

Main features include the digitized versions of the Balocco Proving Ground ride comfort roads, an advanced formulation of bushing/mounts behavior (called handling ride harshness – HRH) for metal/rubber mounts and for hydromounts, a formulation for frequency selective damping (FSD) shock absorbers, and NoSpin tires. It also includes a specific post-processing module able to calculate the standard metrics that FCA uses to diagnose and assess engine shake (called DiffNorm).

Idling models

The idle condition consists of a stationary vehicle, in which the excitation of engine itself is of relatively low amplitude and

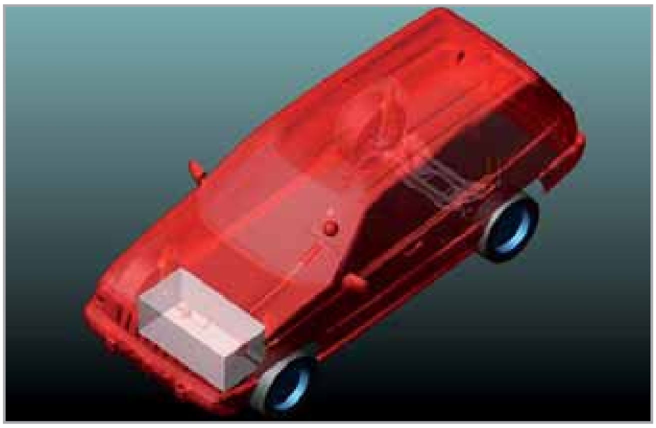


Fig. 5 - Idling model

at specific frequencies (engine orders). In this condition, the hysteresis of the dampers hinders the vertical motion of the suspension.

The model used to represent this condition was a complete vehicle with NoSpin tires, locked suspension, and mounts operating in their dynamic stiffness and damping state, which was calculated using the amplitude and frequency of the vibrations during idling (Fig.5).

Modal analysis

In order to relate the idling performance to the corresponding modal map, a modal analysis was performed in idle condition for each configuration.

MB-Sharc post-processing allows kinetic energy distribution to be identified for each rigid part and for each mode. We can see the frequency and distribution of energy for each direction for each mode of the powertrain (Fig. 6).

Dynamic analysis

The idling performance is expressed in terms of the vibrations transmitted to the car body. Inputs were calculated for a three-cylinder engine, without counter-shafts and with FCA's specific mass balance on the crankshaft. These were then dynamically applied to the engine center. The inputs took into account combustion and the reciprocating masses, as well as the unbalanced rotational forces and a statistically generated ignition irregularity effect.

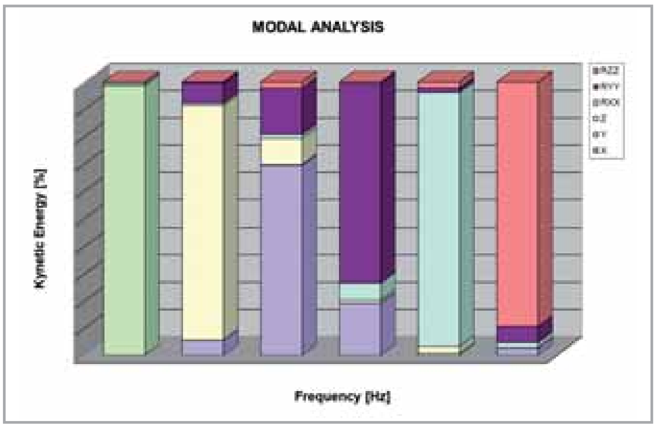


Fig. 6 - Modal analysis

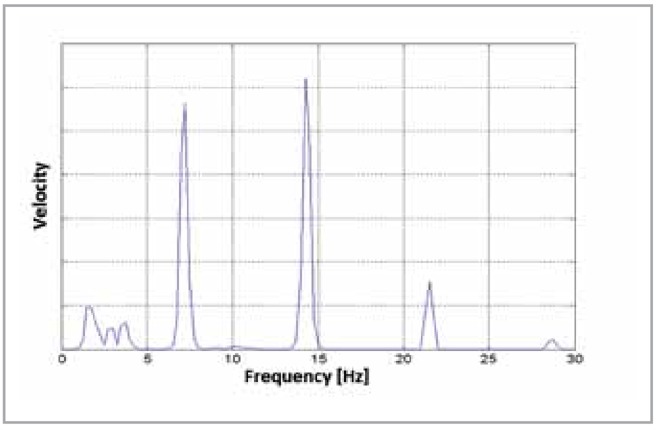


Fig. 7 - Seat rail velocity FFT

The output is the Fast Fourier Transform (FFT) value for the velocity at the seat rail, and the peak values measured at the frequencies of the orders 0.5, 1 and 1.5 (Fig.7).

Ride comfort model

A vibrational model of a complete vehicle was used to simulate a ride comfort track. Acceleration sensors were placed in the correct positions and were used to calculate the DiffNorm diagnostic parameter developed by FCA, in order to isolate the effect of engine shake from all the other sources of vibrational discomfort and evaluate it (Fig.8).

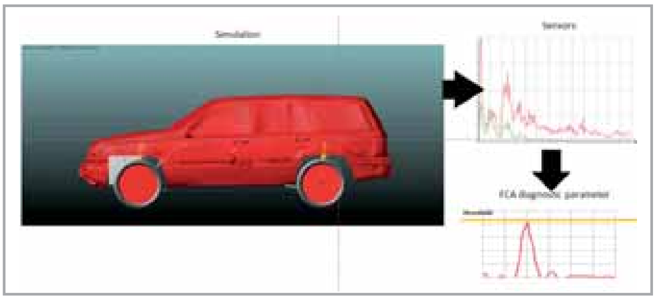


Fig. 8 - Engine shake assessment

This model had already been validated in the normal configuration across a full range of ride comfort simulations.

modeFRONTIER® workflow

A modeFRONTIER® workflow was created to make all the models presented above work together.

There are seven input variables: three static stiffness values for the right mount (one for each direction), three static stiffness values for the left mount, and one static stiffness value for the torque rod. Each stiffness value is then transformed into the correct dynamic stiffness value, depending on the simulation to be performed. Stiffness constraints in the different directions were imposed in order to avoid unworkable mounting designs.

The workflow includes the modal and dynamic analyses for the idling state and the ride comfort analysis for engine shake. For each design, the DiffNorm (engine shake), and the three idling peaks at the orders of 0.5, 1, and 1.5 were calculated. The results of the modal analysis are stored for each design, in order to compare them to the dynamic performance during idling. An activity flow chart is presented in Fig. 9.

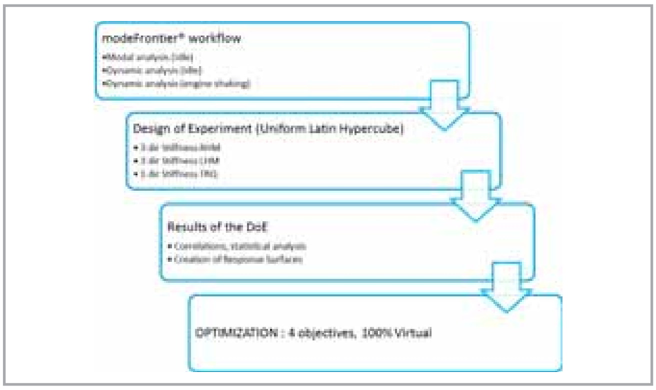


Fig. 9 - Flow chart of the modeFRONTIER workflow

Design of experiments

Based on a uniform Latin Hypercube, a DoE of 200 designs was launched. The results served to create an initial view of the problem, and allowed us to better understand the link between the modal analysis and the dynamic response during idling.

Simple but powerful observations can be made by plotting the modal frequencies against the maximum peaks during idling. Fig.10 shows that the best performance (i.e. the lowest vibrations) is obtained if the yaw frequency is higher than the first order input

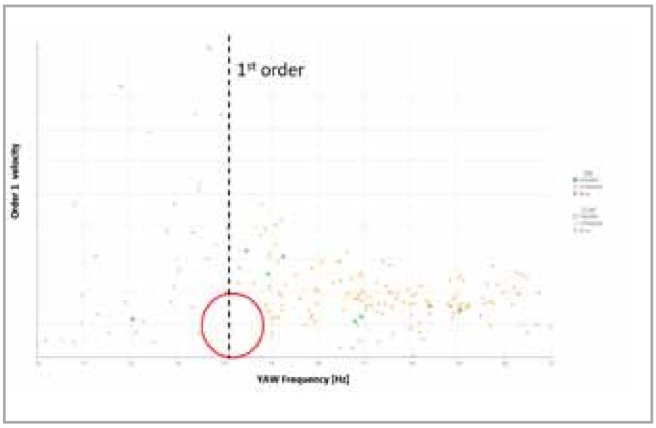


Fig. 10 - Yaw Frequency vs First Order Idling

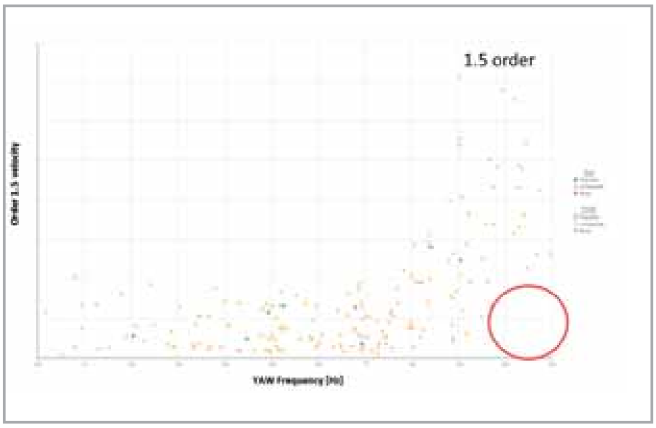


Fig. 11 - Yaw frequency vs order 1.5 – Idling engine

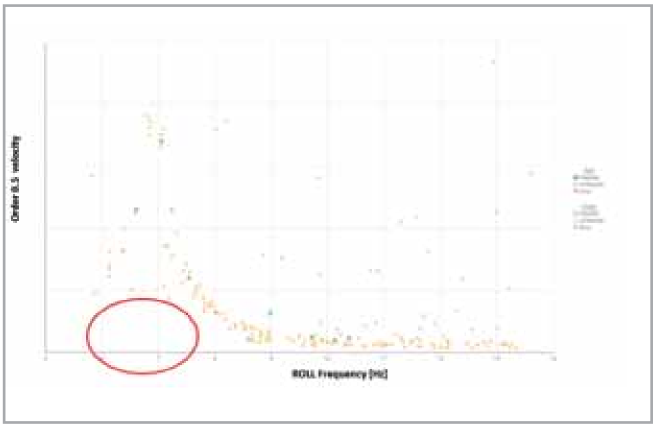


Fig. 12 - Roll frequency vs order 0.5 – Idling engine

frequency. Low peak values with higher dispersion can also be obtained if the yaw frequency is lower than the first order. If the two values are too similar, it is impossible to find a good solution (red circle).

Similar behavior can be observed for other order/mode pairs, if chosen correctly, demonstrating the solidity of the workflow and validating the strategies used to set the stiffness of the powertrain's suspension. As examples, see Fig.11 (yaw vs order 1.5) and Fig.12 (roll vs order 0.5).

RSM

Response surfaces are multidimensional mathematical functions, which are generated based on the results of the preliminary DoE.
For each of the four outputs (the DiffNorm and the three idling peaks) we tried to find the best function of the seven stiffness input values.

In Fig.13, two three-dimensional sections of the surfaces are shown. The validation of the surfaces is done by comparing their value with those of the DoE configurations not used for their training.(Fig.14).

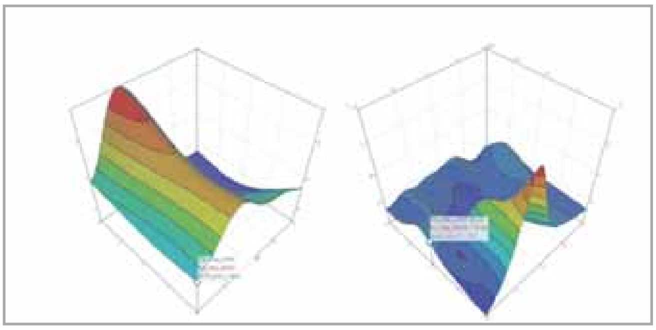


Fig. 13 - Three-dimensional section of some of the response surfaces

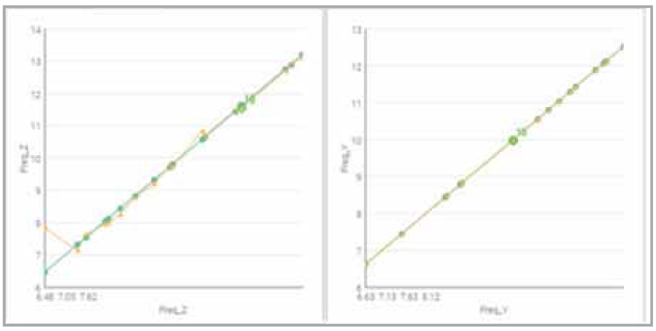


Fig. 14 - RSM validation

Performance	Engine Shaking	0.5 order idle	1 order idle	1.5 order idle
Improvement wrt BASELINE	8.6%	60.2%	45.9%	78.0%

Fig. 15 - Optimization improvements

	Freq X	Freq Y	Freq Z	Rx	Ry	Rz
Position wrt 1st order	Lower	Lower	Lower	Higher	Lower	Higher

Fig. 16 - Modes vs 1st order, optimal solution

Optimization

The creation of the response surfaces allows a very rapid optimization phase. A genetic algorithm (NSGA-2) was used. After identifying the Pareto frontier, the most promising designs on the frontier are selected and validated on the multibody models. Fig.15 shows the results of the optimization, in terms of improvements compared to the baseline design.

As can be seen, the optimal design (even run on multibody models) achieves performance improvements in all areas, particularly for the idling vibrations.

If we examine the modal frequencies with reference to the first order frequency (Fig.16), we see that Rx and Rz are higher. This means that, at some engine speed above idle condition, there may be a correspondence between engine order and resonance, particularly for yaw, which must be carefully evaluated.

Conclusions

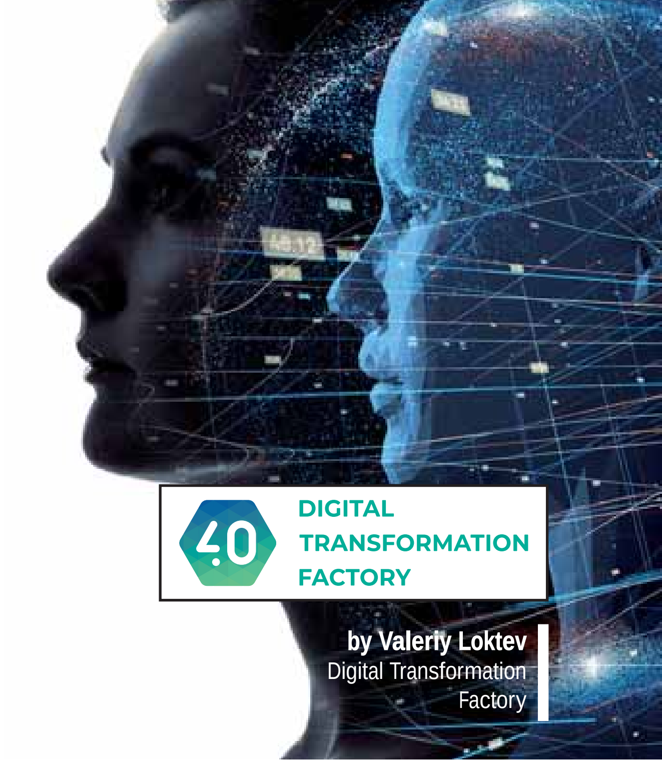
We created a workflow in modeFRONTIER® to evaluate the effect of stiffness of the engine suspension in relation to idling and ride comfort. The workflow included state-of-the-art multibody models developed in MB-Sharc, the customization of Adam/Car used by Fiat Chrysler Automobiles. By means of a first DoE, a set of response surfaces was trained.

A virtual optimization on these response surfaces found an optimal solution that, validated on the multi-body model, proved to be extremely beneficial for both engine shake and idling. The modal map of this optimal design, however, was different from the baseline in terms of resonance frequencies that were higher or lower than the orders of the idle condition.
For this reason, further evaluations are underway to decide whether the optimal design is robust enough for all other constraints and performance targets.

For more information
Stefano Pizzuto - Fiat Chrysler Automobiles
stefano.pizzuto@fcagroup.com

Creation of an Ultimate Digital Twin for a real-world environment

Establishing a simulation-based Digital Twin to evaluate the technical condition and predictive maintenance of power plant equipment



by Valeriy Loktev
Digital Transformation Factory

In this article for business decision makers, DTF explains, in layman's terms, the steps involved in actually creating a Digital Twin for a real-world environment, using the example of a customer that has a thermoelectric power plant used for the production of steel. The business objectives of the customer are presented and then the article describes how a digital twin can be constructed to support those business objectives, providing an overview of the types of technologies, data collection and processing and mathematical modelling required.

How does one define a mathematical model for a production system that can predict malfunctions and enable improvements in efficiency and quality? Such a model would have to contain complete information about each component, including data about the characteristics of the parts and assemblies, the engineering systems, automation devices, work life, maintenance periods, etc. It should also include detailed descriptions of the physical and chemical processes, energy consumption and production processes, as well as information about the input products and raw materials arriving at the plant.

Up until recently, if one tried to do this with traditional technologies, it would create a highly complex mathematical model that would require vast computing resources to simulate the effects on the system of changing any of the internal or external conditions. This complexity would not allow the model to be used in real time, reducing its relevance to the real business assets.

The promise of modern and emerging technology is such that, today, the world's largest companies are investing time and energy on achieving this vision, because enabling the optimal and uninterrupted operation of equipment is the most critical factor in achieving productivity gains, cost-efficiency and improved product quality. As an example, consider a metallurgical company that owns its own thermoelectric power plants to produce steel. One of its largest plants is located in Western Siberia and has seven power units. This plant generates over 500MW of electricity and 1,300Gcal/h of heat, and supplies heat and electricity not only to the metallurgical complex but also to a nearby city. The company decided to create a predictive diagnostics system using Digital Twin technology to guarantee total control of the operation of their key assets, the assessment of their condition, and the prediction of anomalies.

One of the plant's most important assets is a steam-powered turbo generator. It provides electricity and must operate reliably to support the company's continuous work on major metallurgical processes. Operational statistics revealed that there were some defects that could interrupt the generator's operation or significantly reduce its efficiency, which would increase costs from downtime and repair. In assisting the customer to create this portion of the Digital Twin, DTF's consultants studied the SCADA statistics and expert testimonies to identify how faults occur and what parameter deviations preceded them.

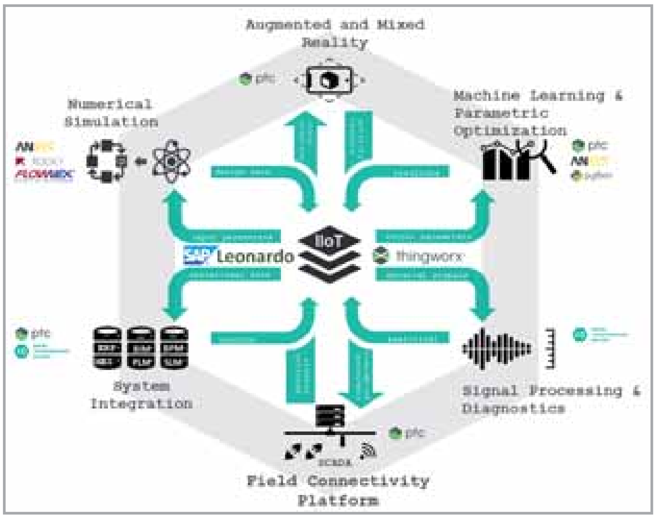


Fig. 1 - Architecture of Digital Twin Solution

Another fundamental asset to the company is the mill, which prepares the fuel that is burned to produce steam in the boiler and therefore must also operate optimally and reliably. The customer requested recommendations on the optimal operating parameters for the mill and also wanted to be able to anticipate certain operation defects in their initial stages.

The company's third vital system is the exhaust, which has sometimes been the weak point of the power plant due to defects in the bearings, shafts, and foundations. The company therefore wanted to predict the worklife of the bearings. In addition to monitoring these three assets, the company also had some other problems in the power plant, such as fouling in the heat exchangers. Fouling or dirt on the heat transfer surfaces of heat exchangers and condensers significantly negatively impacts the turbo generators' operational efficiency. The Digital Twin had to help solve this critical problem by calculating the extent of the fouling and by forecasting its spread.

The Digital Twins makes use of Industry 4.0 technologies to help solve these problems. Digital Transformation Factory (DTF) advocates a hybrid approach that leverages machine learning technologies for those processes that are difficult to describe, but are fairly consistent, and the use of simulation system models as a reference for processes that may vary considerably.

The key element of a hybrid approach is an Industrial Internet of Things (IIoT) platform such as PTC Thingworx or SAP Leonardo. This is a single environment that collects data from an online metrological digital control system, data streams from an automated process control system, and statistics and reference data from related IT production systems, and accumulates them into a single structured database, and provides analytics and visualization.

The key features of the Digital Twin include:

- continuous monitoring of the technical condition of equipment and processes;
- identifying the factors that lead to technological process disruption and defects at early stages;
- identifying changes in the technical condition of operational equipment resulting from one or more defects;
- identifying existing and developing defects in equipment and prioritizing them according to potential damage and residual life;
- advising on how to localize and prevent the development of defects; generating reports on anomalies identified in the operation of equipment, giving reasons; automatically notifying users of pre-emergency conditions

It is important to understand the methodology that allows the Digital Twin to reliably predict faults and abnormal processes in assets by simulating the physical processes and leveraging machine learning. IIoT platform tools support data acquisition and basic signal processing techniques, which facilitate the acquisition and analysis of high-frequency, high-fidelity signals from a variety of sources.

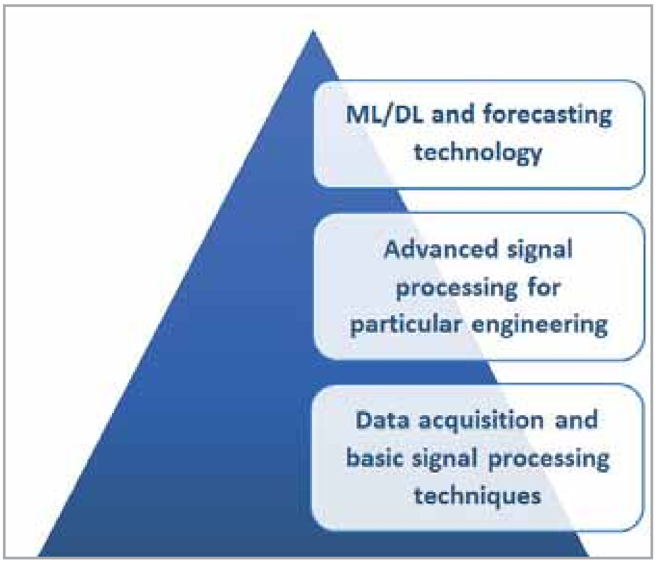


Fig. 2 - Background Technology Basics of Hybrid Digital Twin

All the function of a powerful IIoT platform conduct data acquisition and basic signal processing, including:

- Data acquisition from EC&I level;
- Quantification and synchronization of different signals;
- Using CAE models to provide time-series data that is impossible to obtain physically;
- Conducting basic signal processing to process raw data and conduct initial evaluations of the required data, such as:
  - Noise assessment and reduction;
  - Signal filtering;
  - Spectrum analysis;

Statistical measurements: mean and median, trend evaluation, etc. The second phase of data processing concerns advanced signal processing for specific engineering tasks. This can be done inside the IIoT platform or outside in separate software and includes:

- Data processing to evaluate current load conditions and status;
- Data preparation for offline or online machine learning / deep learning (ML/DL) and forecasting (which cannot be done with raw unprocessed data);
- Advanced processing techniques using statistical or mathematical methods to forecast residual life or make reliability measurements.

ML/DL and forecasting technology are used to substitute the processed data with surrogate mathematical data models for rapid response evaluation. Several tools can be used to generate surrogate data models:

- In some cases, it can be done inside the IIoT platform (e.g. PTC ThingWorx Analytics);
- Alternatively, it can be done using advanced metamodeling approaches in ANSYS optiSLang with additional connections to the IIoT platform.

Preparing the data for forecasting is an advanced process that involves different logic for different engineering tasks. It can also

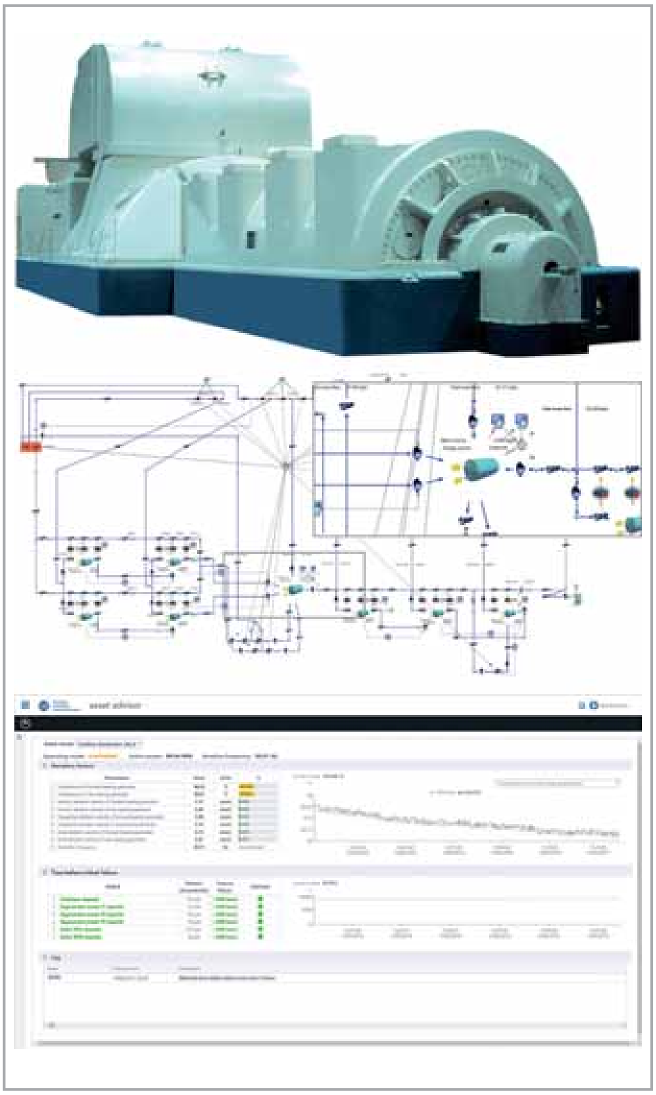


Fig. 3 - Evaluating the effects of defects in an unit on power plant efficiency

be called signal processing because it has to prepare the data for the ML/DL and forecasting technology. These are used with 1D and 3D modelling and optimization, which are widely used for design projects, and can also be used during operation.

Various mathematical tools are used in the Digital Twin:

- System simulation. The real-time system model simulates the physical phenomena that occur under actual operating conditions in the asset. The effectiveness of the simulation tools is confirmed by industry certificates (ASME, NQA1).
- Multidisciplinary 3D simulation. Simulation results are implemented in the system model by reducing the order of the 3D model, which ensures the Digital Twin's high accuracy and efficiency.
- Automated verification. Resolving identified parameter problems for mathematical models with hundreds of variables ensures that they correspond to the real state of the asset under production conditions and that they are analytically accurate.
- Multi-objective parametric optimization. The search for optimal operating parameters is based on previously developed meta-models. Recommendations are issued in seconds and are followed by 3D visualizations of the physical processes.
- Machine learning. Early detection of defects is performed at two levels, firstly by ensuring that changes in the signals of individual sensors do not exceed the "normal" production parameters, and secondly by using machine learning models to study complex deviations.

There are three main approaches to building the analytical core of a Digital Twin, but in common practice we have found it most effective to combine them. These are:

1. Direct coupling with CAE. This approach provides maximum fidelity and validity of results, but the simulation models requires a long time to set up, verify and calibrate. In addition, this approach cannot be used directly in pseudo real-time because of the long computing times required, which is why this approach cannot be used as is. Although ANSYS is shown in the figure, it is used to provide specific low-fidelity codes that can provide a quick solution in a very short time. We do use special metamodeling techniques to obtain fast surrogate metamodels, which can be used in pseudo-real time by digital twins.
2. 0D system modeling techniques or direct mathematical models development using Python, MATLAB, etc. This approach can be successfully used in pseudo real-time by digital twins and provides sufficient result fidelity and validity in most cases, but it is quite difficult to develop a universal numerical model of a device or process with a wide range of operating modes and conditions.
3. Unique metamodeling technique. This differs from the standard Reduced Order Modelling (ROM) approach by using more complicated algorithms and providing greater flexibility. The resulting metamodel can be connected directly to the IIoT-platform or embedded inside the system model as a high-quality ROM. However, this approach requires the use of a lot of raw unprocessed data in the metamodeling to achieve greater predictive accuracy. It is also very difficult to keep this model effective in cases where the real object changes over time (e.g. material properties change, wear and tear, etc.).

There is no unique best approach or 'How to' for Digital Twins. Each application requires a careful analysis of the situation and the customer's requirements. We generally find that combining all of these approaches generates the best results.

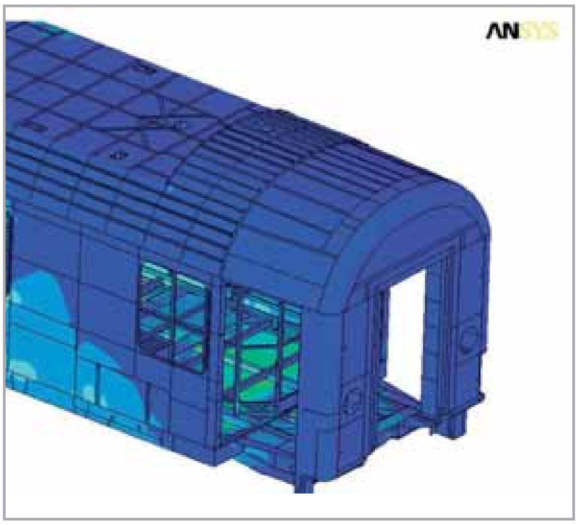
For more information
Valeriy Loktev - Digital Transformation
valeriy.loktev@digitaltwin.ru

Leading the Way in Land Transport

ANSYS in the Design Process

The MERMEC Group, a world leader and innovator specializing in providing integrated diagnostic solutions and the efficient management of rail, metro and tramway infrastructures, has selected ANSYS Mechanical simulation technology for its Technical Department as the group gears up for Industry 4.0. The Group has made innovation and continuous investment in Research & Development its mission, becoming the only supplier of its type in the world capable of designing, developing and manufacturing in-house all the solutions in its product portfolio. Fifteen percent of annual turnover is invested in research activities.

Established as a public company in 1988, MERMEC employs more than 500 highly specialized professionals located in 17 countries. The product and service offering are organized into five strategic business areas: systems and vehicles for diagnostics of infrastructure and rolling stock, new generation railway signaling solutions, software platforms for optimizing and planning railway infrastructure maintenance and renewal operations, professional services, and diagnostic systems for the steel industry.



MERMEC's technical management examined and finally chose ANSYS Mechanical simulation technology because it considered the Workbench platform to be the best suited for significantly increasing its Technical Department's productivity, which is preparing to decisively address the new design challenges posed by Industry 4.0.

"MERMEC's philosophy has always been to invest in the best technologies," said Dr. Chiara Pertosa, CFO of MERMEC, *"It was with this philosophy in mind that we chose ANSYS because we believe it to be the best solution for our needs."*

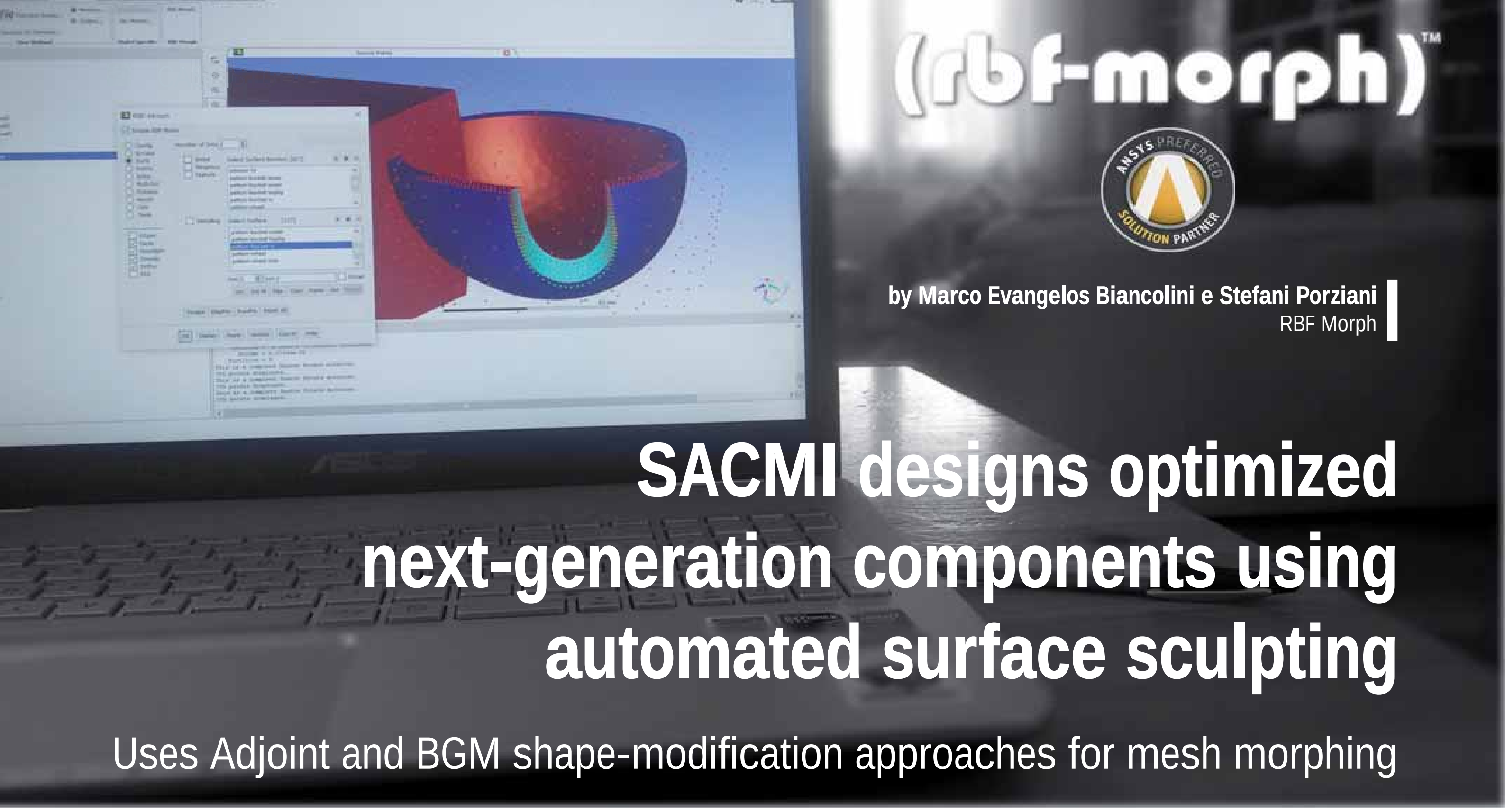
Given the increasing number of case studies that it needs to analyze structurally, MERMEC chose to work with the ANASYS Workbench environment because it enables them to define intuitive workflows, carry out quality thermo-structural analyses in appropriate timeframes and, finally, to create analysis "templates" that can easily be reused for other similar analyses, but using different geometries.

"The ease-of-use of the latest versions of ANSYS Workbench further convinced us that we had made the right choice by moving towards ANSYS solutions," continued Vito Valentino, engineer and head of the Technical Department at MERMEC. *"EnginSoft has also proven to be a serious and reliable partner by helping us with practical operational support for our specific problems in the start-up phase, which is always very delicate."*

With regard to geometries, the possibility of directly integrating the models developed into an external CAD environment was particularly helpful. The direct interface tool to the CAD environment allows all geometric parameters to be imported into ANSYS Workbench so that they can then be used in the multi-objective optimization loop after the completion of the analysis.

The projects developed by MERMEC often consist of complex systems made up of large-scale metal structures; the designer and the analyst are therefore required to define their behavior from both a global and a local perspective. In these cases, ANSYS Workbench enabled a drastic reduction in calculation times to be achieved using the sub-modelling technique, which allows MERMEC to study both the global stress conditions over the entire assembly, and to obtain top quality results when they automatically switch to the detailed model.

"Overall, we are satisfied with the choice we made which will allow us to increase the quality of our products and drastically reduce design times," concluded Valentino.



by Marco Evangelos Biancolini e Stefani Porziani
RBF Morph

SACMI designs optimized next-generation components using automated surface sculpting

Uses Adjoint and BGM shape-modification approaches for mesh morphing

Mesh morphing has proven to be a valuable tool in parametrizing numerical models to perform shape optimization. It allows engineers to save time in generating new configurations for analysis because it does not require geometry modification and mesh re-generation. However, engineers still have to estimate which shape modifications could improve component performance and then apply them to the baseline model before resolution. With automatic surface sculpting, the morphing action can be guided by the physics of the model that is being analyzed, allowing engineers to automatically identify the surface zones whose nodes should be moved inwards or outwards to achieve the optimization objective, saving on the time required to complete an optimization process.

In industrial production, design optimization can be the key to a product's success. Through optimization a company can maximize profits and minimize both costs and the use of raw materials. Optimization can be a very challenging task for engineers, however, since the optimal configuration for a product must be sought while maintaining its compliance with the task for which it was designed (e.g. strength and functionality requirements). Numerical simulations, both computational structural mechanics (CSM) and computational fluid dynamics (CFD), can help engineers to test different configurations to identify the optimal one. Each numerical model represents a different product configuration that, once analyzed numerically, provides relevant information on performance indices.

Although numerical procedures have brought considerable benefits, a single numerical simulation, depending on the size of the model, the complexity and the type of analysis required, can take a long time to generate the data desired. As is well known, each numerical analysis consists of three main phases: pre-processing, in which the numerical model is created and set-up, resolution, in which the numerical solution is evaluated, and post-processing, in which the numerical results are examined and analyzed.

To shorten the time required to complete the numerical analysis, improvements can be made to the first and second phases: High-Performance Computing (HPC) allows the resolution time to be reduced, whilst mesh morphing enables considerable time savings during pre-processing. Mesh morphing allows engineers

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

to generate a new configuration for the numerical model without the need to generate the modified geometry and without the need to regenerate the computational grid. This latter task can be really time consuming in an optimization study especially for high-fidelity meshes since it has to be repeated for each configuration to be analyzed (see Fig. 1).

The RBF Morph™ morphing tool has proven its effectiveness and reliability in performing this task, generating time savings of up to 80% in pre-processing while producing high-quality morphed meshes with minimal mesh distortion.

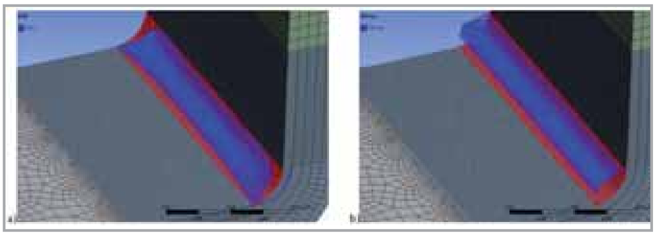


Fig. 2 - Comparison between a) free and b) filtered BGM surface sculpting

	A	B	C	D	E	F	G	H	I
1	Name	Update Order	[illegible]	[illegible]	[illegible]	[illegible]	Ret...	Retained Data	Note
2	Units			Pa	Pa	MPa			
3	DP 0 (Current)	1	0	[illegible]	[illegible]	[illegible]		✓	
4	DP 1	2	1	[illegible]	[illegible]	[illegible]		✓	
5	DP 2	3	2	[illegible]	[illegible]	[illegible]		✓	
6	DP 3	4	3	[illegible]	[illegible]	[illegible]		✓	
7	DP 4	5	4	[illegible]	[illegible]	[illegible]		✓	
8	DP 5	6	5	[illegible]	[illegible]	[illegible]		✓	
9	DP 6	7	6	[illegible]	[illegible]	[illegible]		✓	
10	DP 7	8	7	[illegible]	[illegible]	[illegible]		✓	
11	DP 8	9	8	[illegible]	[illegible]	[illegible]		✓	
12	DP 9	10	9	[illegible]	[illegible]	[illegible]		✓	
13	DP 10	11	10	[illegible]	[illegible]	[illegible]		✓	
*									

Fig. 3 - Parameters set-up in BGM sequential analysis

Surface sculpting using mesh morphing with BGM

Mesh morphing also offers the ability to guide the modification of the shape of the numerical model using the physics of the model itself. The morpher tool described next is the RBF Morph™ ACT extension for the ANSYS® Mechanical™ environment. The software was released in 2015 and is based on the ANSYS® Fluent™ Add On technology that has been available on the market since 2009 (www.rbf-morph.com). Since the first version, several features have been added to help engineers in their numerical simulation activities. One of the latest features introduced is the Biological Growth Method (BGM) approach for surface sculpting. The BGM was inspired by the observation of the behavior of biological tissues: in nature, the number of tissues increases if the stresses on the surfaces reach an activation value, which changes with the type of tissue.

The same approach can be applied to structural components. Furthermore, material layers are added if the stress levels are above a threshold value, and eliminated when the stresses are below the threshold.

Using the BGM approach for surface sculpting has two advantages: the maximum peak stresses can be reduced and the stress distribution in the model becomes more uniform. However, since the surface sculpting is guided by the stress distribution, the resulting shape can be very complex, and can only be created on the real physical object by using innovative additive manufacturing technologies.

If the manufacturing process concerned is a traditional one (such as casting, machining, etc.), RBF Morph offers an additional function that allows engineers to respect their manufacturing constraints. The coordinate filtering option allows engineers to replicate the modification of the shape in a specific position (e.g. the position in which the maximum stress was evaluated) and to apply it along a user-defined axis (Fig.2). Both Cartesian and cylindrical coordinates can be used as filters.

The highest degree of automation can be achieved by linking RBF Morph surface sculpting based on BGM with the definition of the parameter-based Design Point in ANSYS® Workbench™. The analysis can be set up so that, after solving the baseline configuration (DP 0), the analysis results are used to perform the surface sculpting. The subsequent Design Point (DP 1) is generated from the morphed configuration of the previous DP.

That is then resolved and the evaluated stress distribution is used to set up another surface sculpting step. Using Workbench parameters, this procedure can be repeated until the stress levels reach the desired optimal values, as shown in Fig.3. For example, it is possible to consider the problem of a cantilevered beam: beam theory tells us that the optimal configuration for this simple static structure has a parabolic profile. If BGM sculpting is performed on a beam of constant thickness where the upper and lower surfaces are those to be sculpted, it is possible to automatically reach the shape depicted in Fig.4, which has a parabolic profile, a uniform stress distribution and a reduction in mass of 33% compared to the initial configuration.

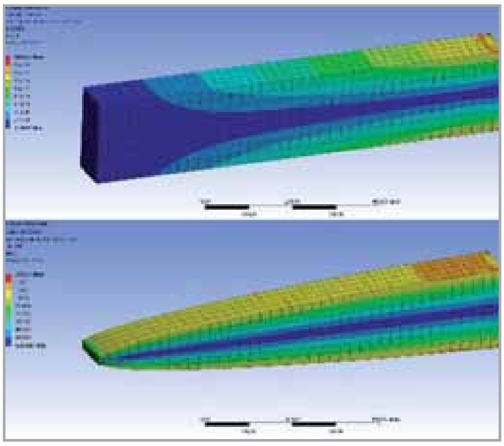


Fig. 4 - BGM surface sculpting applied to a cantilevered beam

Surface sculpting using mesh morphing with the Adjoint approach

A second approach to surface sculpting with RBF Morph mesh morphing is the Adjoint approach. The Adjoint method is widely used in CFD simulations and has recently also been introduced in CSM problems. The output of the Adjoint solution consists of sensitivity to a defined objective function of a set of input parameters. If nodal surface displacements are set as input parameters, with the Adjoint method engineers can determine which surface node should be moved inwards or outwards. This information can be transferred to the RBF Morph set-up so that an automatic surface sculpting approach can be

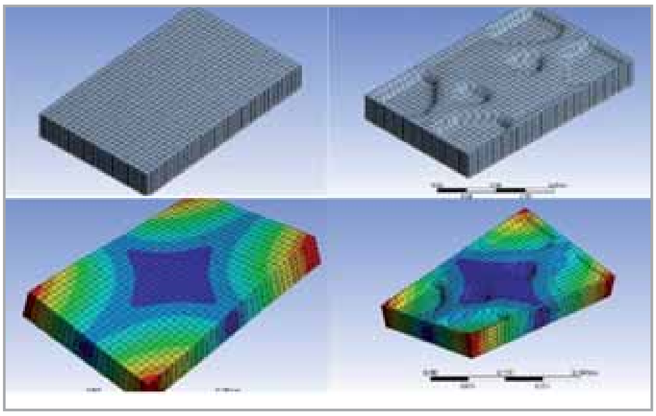


Fig. 5 - Thick plate optimization with Adjoint sculpting

used. With the Adjoint approach, it is possible to define and use several objective functions as an optimization objective: not only stress level reduction (as in the BGM approach), but also mass reduction, compliance, and so on. For example, the optimization of a thick plate is shown in Fig.5. The free and undamped modes of the plate are extracted using an eigenvalue analysis. The optimization task aims to reduce the mass of the component by keeping the first eigenvalue above 1220Hz. Adjoint data was used to set up the mesh morphing, resulting in the optimized shape shown in Fig.5, which produced an initial eigenvalue of 1351.5Hz and a 20% mass reduction.

Applications for surface sculpting

SACMI, an early adopter of the RBF Morph optimization since 2015, successfully used automated surface sculpting procedures in an exploratory study. The Italian company SACMI Imola S.C. is an international leader in the design of industrial plants. It supplies both machines and complete systems for the ceramics, metals, packaging, and food and beverage industries, and for the production of containers made of plastic and advanced materials. Due to the confidential industrial nature of the component analyzed (Fig.6), it is not possible to provide detailed data on the model. However, both automatic sculpting approaches were used to optimize the component's stress levels. The Adjoint-based approach was successfully applied to the area identified in Fig.7a and resulted in a maximum stress reduction of 11.6% on the surface shown in Fig.7b.

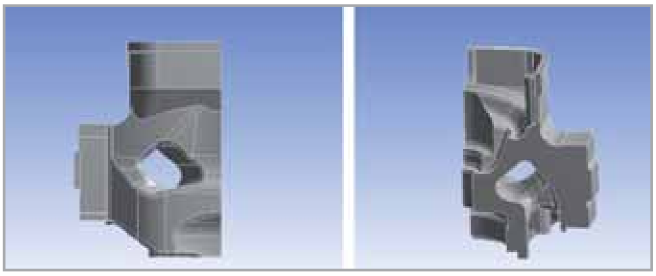


Fig. 6 - SACMI industrial component

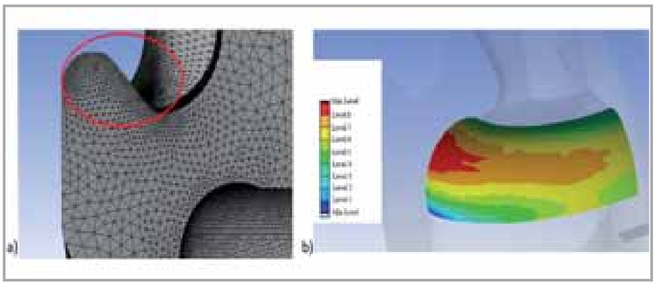


Fig.7 - a) Area in which Adjoint-based sculpting was applied, b) resulting stress distribution

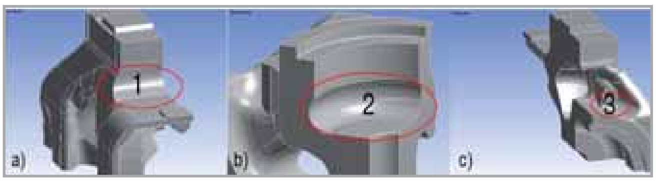


Fig.8 - Optimization of SACMI industrial component hot spots

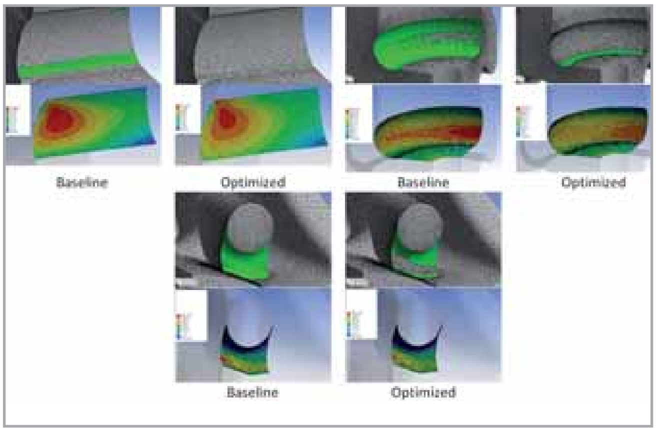


Fig. 9 - BGM-based sculpting on SACMI component hotspots

The BGM approach was applied to three zones of the component (Fig.8), using the linear manufacturing constraint in hot spot area 1, the circular manufacturing constraint in hot spot area 2, and leaving the BGM algorithm free to sculpt the surface in hot spot 3 area.

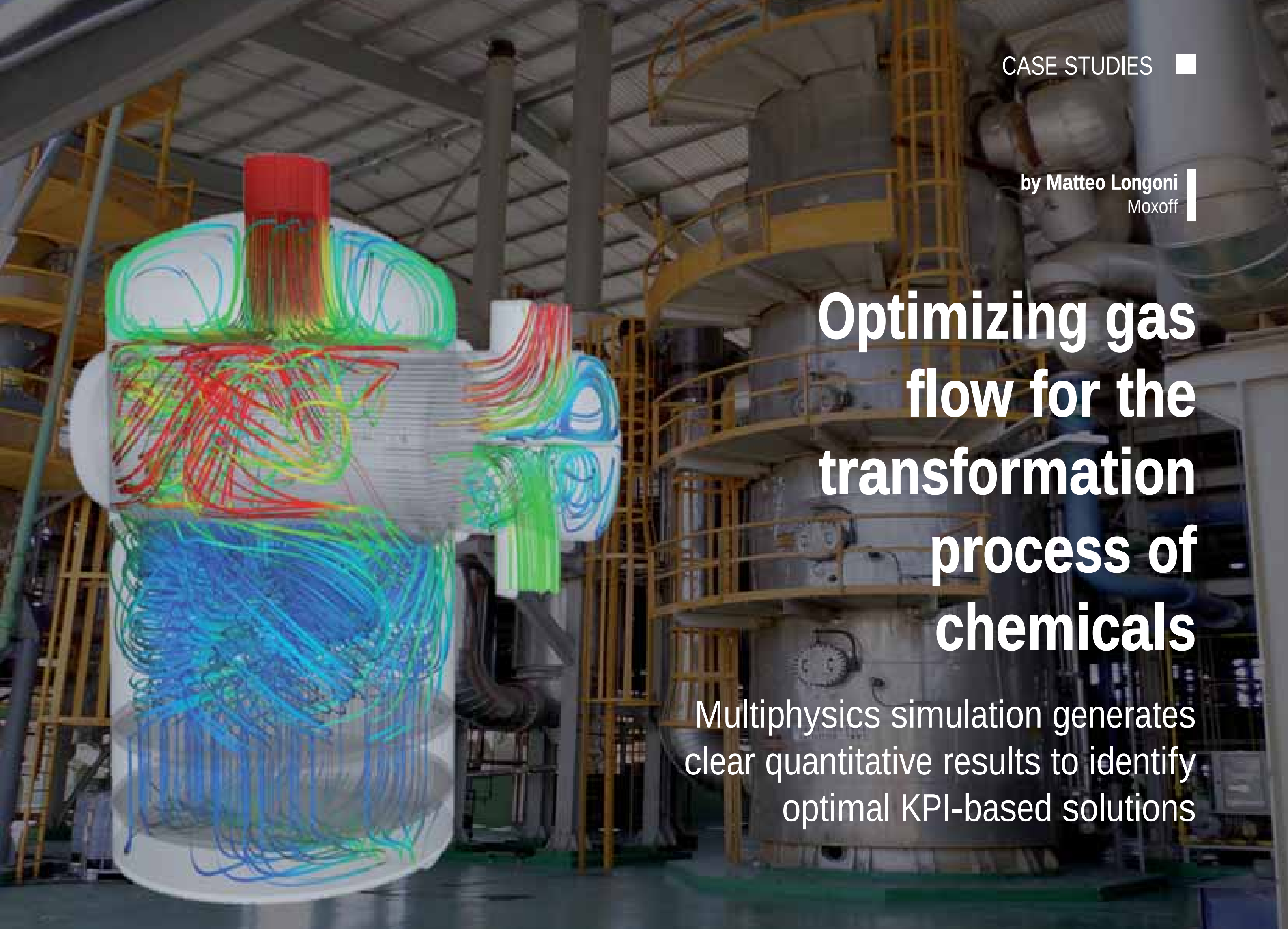
After 8 sequential steps, the maximum surface stress was reduced by 2.76% for hot spot 1, by 7.84% for hot spot 2, and by 8.12% for hot spot 3. Fig.9 shows the baseline and final meshes respectively, and the initial and final surface stress distributions for the three hot spots.

Conclusions

The study presented aims to demonstrate how to perform product optimization automatically, saving on the time required to complete an optimization process. The proposed optimization strategy involves the use of numerical simulations and consists of combining mesh morphing with one of two shape-modification approaches dictated by the physics of the model being analyzed, namely the Adjoint approach or the BGM approach.

The automatic surface sculpting optimization strategy was developed by leveraging the flexibility of ANSYS® Workbench™, within which the RBF Morph™ ACT extension for ANSYS® Mechanical™ allows this task to be easily performed. The surface sculpting capabilities of the morpher tool can easily be set up to be guided by the simulation results in order to automatically identify the model nodes to be moved and to generate a new optimized shape. Workbench's parameter management allows the procedure to be iterated until the desired optimized performance indices are achieved.

For more information
Stefano Porziani - RBF Morph
stefano.porziani@rbf-morph.com



by Matteo Longoni
Moxoff

Optimizing gas flow for the transformation process of chemicals

Multiphysics simulation generates clear quantitative results to identify optimal KPI-based solutions

In this technical article, Desmet Ballestra and Moxoff show the benefits of using multiphysics simulation to identify the best configuration for a processing plant for chemicals and detergents, according to customer-defined key performance indicators (KPIs). Simulation enables clearer quantitative results and therefore, faster, and more cost-efficient evaluation of highly complex and hardly accessible systems than laboratory testing and controlled experiments. Furthermore, the innovative simulation approach explained here was able to consider the entire system of this multistage process and then to compare various design alternatives using quantitative KPIs in an iterative optimization process.

The industrial challenge and the simulation approach

Desmet Ballestra is the world leader in developing, engineering and supplying technologies and processing plants for the detergent and chemical industries. Almost every installation is customized to customer specification, and system performance is critical to ensure delivery quality.

In this industrial sector, simulations is necessary when changes have to be made to the configuration of a plant. Simulation is much faster and cheaper than performing laboratory tests, even those conducted on a pilot scale. Moreover, Computational Fluid Dynamics (CFD) simulation enables clearer quantitative results in a highly complex and possibly confused system – even controlled experiments cannot deliver equally good quantitative results.

This article presents a project to optimize gas flow in the SO2 to SO3 conversion process executed jointly by Desmet Ballestra and Moxoff, a company specialized in developing innovative solutions based on customized mathematical modelling to solve industrial challenges.
In the SO2 to SO3 conversion process, it is essential that as the gas flow reaches the catalytic bed, the flow velocity and the temperatures are homogeneous over the entire section of the conversion tower.

The industrial challenge was to improve the conversion tower's performance by choosing the best configuration among many alternatives and according to several concurring key performance indicators (KPIs). The aim of the project was three-fold: to understand the conversion tower's thermo-fluid dynamics, to identify and compute quantitative KPIs, and to compare alternative technical solutions to maximize the performance of the plant.
Moxoff provided an innovative approach using simulation that took into account the entire system: the multistage process,

the catalytic beds, the heat exchangers and the multiphysics phenomena. Dedicated turbulence models coupled with porous flows were developed in a model tailored to the system being analyzed. Moxoff, in partnership with Omiq, a CFD-specialized company, performed multiphysics simulations over a range of different design and operating conditions, which were then compared and modified using quantitative KPIs in an iterative optimization process.

How a conversion tower functions

The scope of this particular equipment is to oxidize the SO2 stream from a sulfur burner. The exothermic reaction requires a catalyst and is performed in a four-stage adiabatic mode. At the outlet of each stage, the reacting system reaches thermodynamic equilibrium and requires inter-stage cooling to proceed further.
Equipment performances is highly dependent on the homogeneous distribution of temperature and flow across the entire section of the tower. In addition to any turbulence existing at the tower inlet, an additional incorrect distribution may be due to the cross-flow cooling device installed upstream of the first catalytic stage. The main purpose of the project was to design an internal flow equalizer capable of maximizing the gas flow rate and the temperature distributions at the inlet to the catalytic stage.

Thermo-fluid dynamics analysis

The layout of a conversion tower is shown schematically in Fig. 1. The main elements of the tower are the heat exchanger, the catalytic bed, the shell, the conveyor and the two spheres sections.

Boundary conditions (see Fig. 2) were provided by the company, starting from the operational data:

- Fluid on the tube side: air (molecular weight 28.95 Kg/Kmol)
- Fluid on the mantle side: SO2 (molecular weight 31 Kg/Kmol)
- Tube-side outlet (Fig. 2a.): pressure = 104800 Pa
- Tube-side inlet (Fig 2b.): flow rate = 7408 Kg/h, T= 40°C
- Mantle-side inlet (Fig 2c.): flow rate = 14450 Kg/h, T= 650°C
- Pressure after the catalytic bed (Fig 2d.): pressure = 159050 Pa

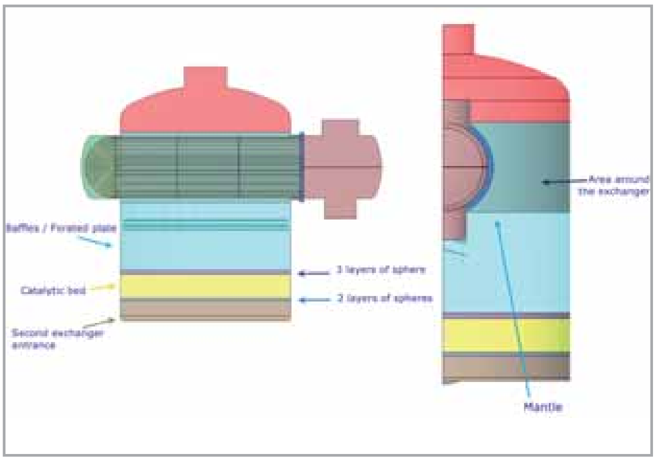


Fig. 1 - Conversion tower diagram

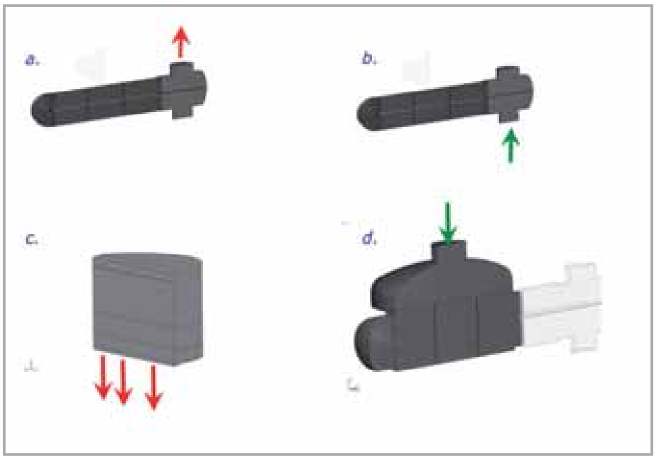


Fig. 2 - Boundary Conditions of the Conversion Tower

- Catalytic bed: fluid volume with porosity = 55%, pressure resistance = 800Pa, thermal source = 158°C
- Two volumes representing the spheres: porosity = 40%, pressure drop = 150 Pa
- Thermal conductivity, thermal capacity and viscosity of the fluids were considered to be variable with temperature.

The fluid and solid 3D geometry of the tower and of the heat exchanger were reconstructed starting from the CAD drawings.
SpaceClaim was used to create and handle the CAD geometries, SimericsMP to generate the mesh and to perform the simulations of the thermo-fluid dynamics and the heat exchanger. Post processing and visualization were performed with EnSight.
To compare the performance of the geometries, a set of KPIs was defined: in particular, the temperature and velocity profiles at the inlet to the catalytic bed of the first stage were analyzed. In fact, temperatures and velocities in these areas are fundamental parameters for evaluating the performance of the reaction that takes place in the underlying catalytic bed. To this end, the first stage exchanger was also modelled, to correctly simulate the flow entering the tower, and thus to achieve a more accurate simulation with more detailed information about the heat exchanger's effect on the flow.

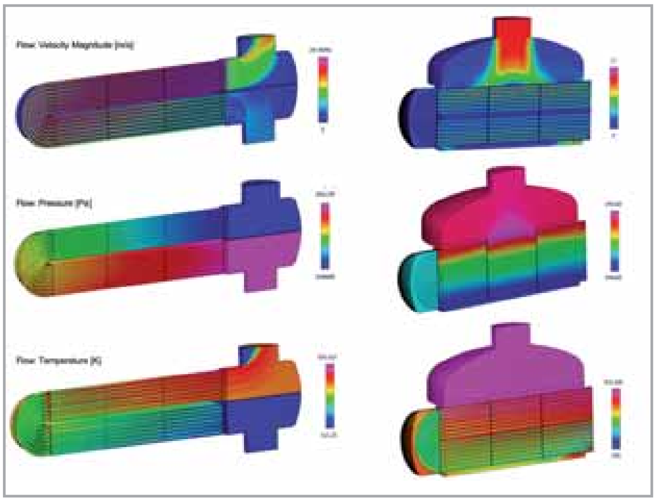


Fig. 3 - Simulations of thermo-fluid dynamics in the heat exchanger on the tube side (left) and mantle side (right)

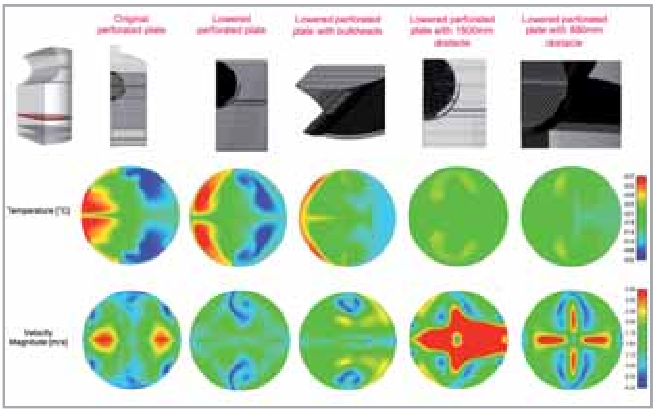


Fig. 4 - Temperature amplitude and Velocity distribution with different configurations of the perforated plate

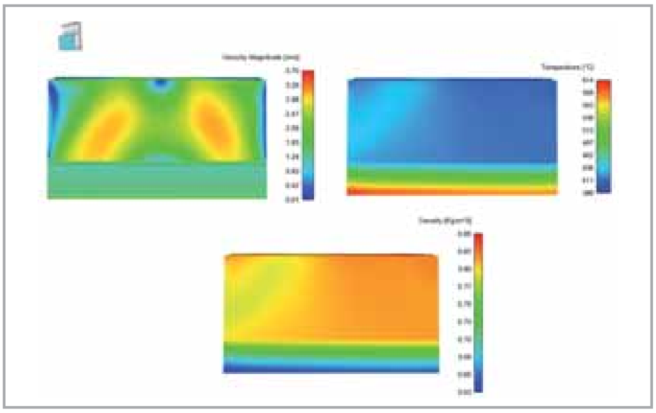


Fig. 5 - Temperature amplitude and Velocity distributions with the original configuration of the perforated plate.

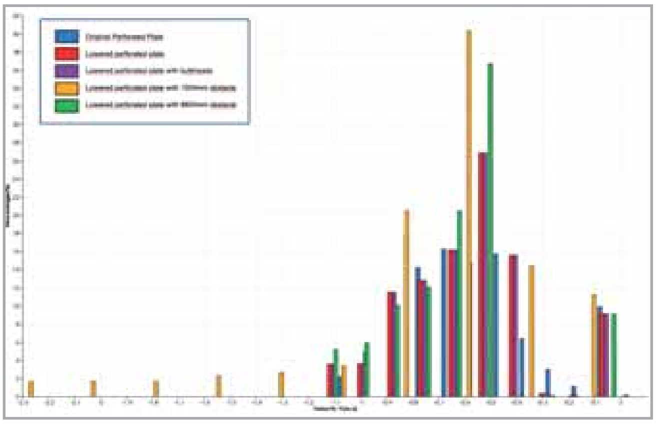


Fig. 6 - Comparison between alternative configurations through a Temperature histogram based on the Temperature area percentage distribution on the catalytic bed.

The mesh was built using different strategies depending on the parts being considered. Thermo-fluid dynamics simulations of the exchanger were run on both the tube side and mantle side. (Fig 3).

Results

Two alternatives scenarios were investigated: a conversion tower with baffles and a conversion tower with a perforated plate. Different configurations were explored for each scenario. We found that the introduction of the perforated plate allowed for better performance: compared to the baffles, which generated asymmetric velocity and temperature profiles on the catalytic bed with peaks in the peripheral zone, the perforated plate dramatically minimized this problem, leading to slower gas flow on average between the mantle and the conveyor, and it also equalized the flow. The flow is carried upwards through a bulkhead towards the recirculation zone of the plate, and then becomes uniform again downstream. Placing the perforated plate at a lower position in the tower enables the number of vortices in the area below the plate to be controlled and reduced. To increase the mixing temperature, and to create a more uniform distribution in the upper spheres section, we inserted two bulkheads and an obstacle at the outlet

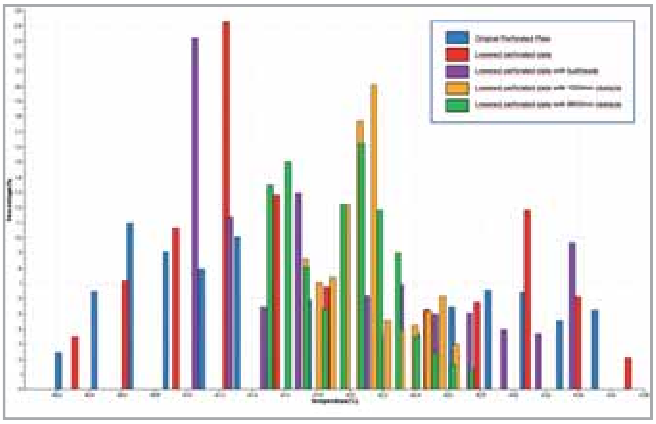


Fig. 7 - Comparison between alternative configurations through a Velocity histogram based on the velocity area percentage distribution on the catalytic bed.

and inside the conveyor respectively. These elements were then sized to ensure the achievement of KPIs such as uniformity of temperature and velocity uniformity in the spherical sections, specifically a total Delta P of about 2000 Pa and a peak velocity at the conveyor outlet of about 40 m/s. Figs. 4-7 summarize some of the simulation results for temperature amplitude and velocity distribution spanning across some of the configurations with the perforated plate. A dynamic exploration of the simulation results is available at [https://youtu.be/0jdeFNfpqDw].

Conclusions

The simulations conducted enabled us to identify the optimal design for the inside of the equipment, based on quantitative and engineering criteria, i.e. a construction that guarantees the achievement of the objectives set and provides the company with an optimization tool for their equipment design process.
Moxoff's multiphysics modelling skills coupled with Desmet Ballestra's industrial know-how enabled the design and manufacturing processes to be orchestrated to best meet the customer's requirements.

For more information
Matteo Longoni - Moxoff
matteo.longoni@moxoff.com

By Nobuhito Nakagawa[1], Taishi Nakamuta[1] and Akiko Kondoh[2]
1. Lion Corporation - 2. Prometech Software, Inc.

Refining the design of a spout for a laundry detergent

Particleworks enables fluid flow simulation combined with human-like behavior to improve product performance

LION Corporation produces health care products such as toothpaste, detergent, soap, etc. The company, which has a long 120-year history, faces a rapidly-changing business environment due to the recent phenomena of an aging society, rising health awareness, and technological demands that affect the planning and development of products and services using digital engineering.

LION's Container and Packaging Engineering Research Laboratories is responsible for designing and developing containers and packaging for all LION products and has used CAE simulation for over 30 years. Their use of CAE is increasing year by year due to business acceleration and diversification, as well as changes in the laboratories. In the past, during conventional product development, the company used a 3D printer to make nearly 100 prototypes of a spout cap design, from which they chose the most suitable design. This time, with the desire to more scientifically verify the designs, with Prometech's cooperation, the company decided to use fluid dynamics simulation with Particleworks, in addition to the 3D printer experiments, for the design of the cap for the new HARETA brand, launched last year. The concept of HARETA, a liquid laundry detergent that contains a polymer to increase its viscosity, is that it always leaves clothing feeling like it has been air-dried on a clear day. The detergent

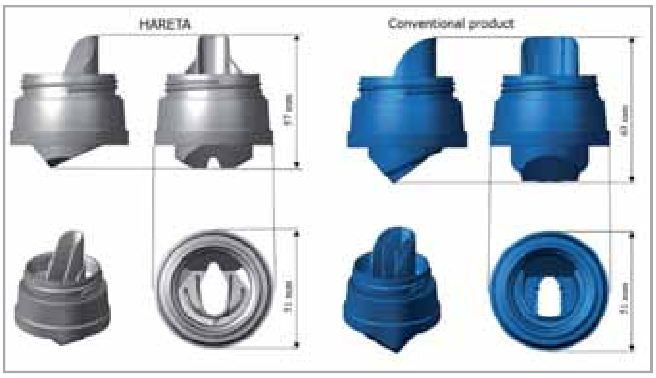


Fig. 1 - Comparison of spout cap design for HARETA and a Conventional product

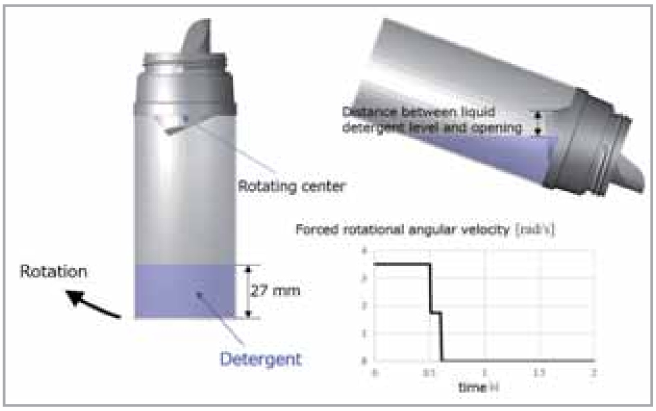


Fig. 2 - Settings of detergent amount and definition of the forced rotation

has a non-Newtonian viscosity meaning that the liquid tends to be less viscous during the pouring process.

Generally, users want to pour the measured dose as quickly as possible, so they unconsciously raise their elbows and tilt the bottle to pour the detergent more quickly. This tendency becomes stronger with high viscosity liquids. This results in the spout becoming blocked by the liquid, so the flow isn't constant. In addition, because the measuring cap is small and the liquid's viscosity is high, the detergent may spill and splash the surroundings. To avoid this, the designers wanted to ensure that the flow-down of the detergent could be measured in a comfortable time. Therefore, they changed the design of the spout cap to improve the detergent's flow-down velocity. Fluid dynamics simulation was performed with the objective of improving the measuring process.

During the Particleworks simulations, a conventional cap model was used to calculate the relationship between the spout cap's configuration and the flow-down velocity in order to have a reference point to compare the flow behavior of the new HARETA designs. The liquid flows from the hole of the cap, moves down through the wall of the spout and falls into the measure. The simulation confirmed that the flow velocity was low as a result of flow resistance near the spout wall, and that the flow velocity increased further away from the wall. Therefore, the designers predicted that the flow-down velocity would be improved by expanding this inner area. Fig. 1 compares the geometry models of the newly developed spout cap for HARETA and a conventional spout cap. Fig. 2 shows the conditions of the flow-down simulation performed using each of these caps. When the detergent is poured, one cannot ignore the air entering the bottle when the liquid surface of the detergent is higher than the opening. This simulation did not specifically consider the influence of air

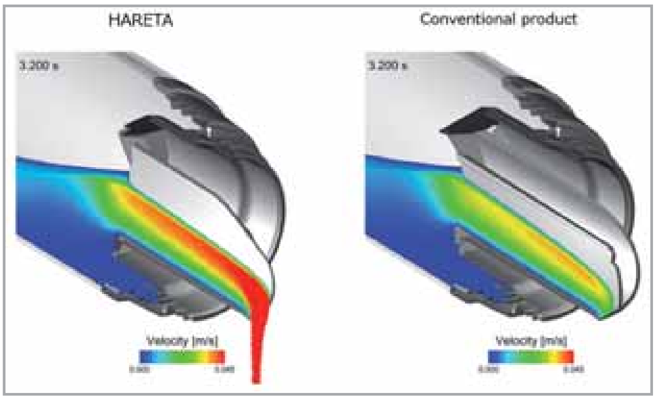


Fig. 3 - Comparison of outflow velocity distribution of detergent in longitudinal section of the HARETA spout cap and the conventional product spout cap

but it defined the detergent amount to not completely fill the opening. In addition, the tilting motion was calculated by applying a forced rotation so that the bottle gradually tilts to 110° between 0 and 0.6 seconds after the start of the pouring motion. As can be seen from the graph in Fig.2, the simulation divided the bottle tilting operation into two steps to reproduce so-called ordinary human movement on the bottle movement as closely as possible. The machine used in LION's experiments can only tilt the bottle at a constant speed, while simulation allows them to reproduce human-like behavior more flexibly. The flow-down simulation was performed to compare the spout cap models of HARETA and the conventional product, using these conditions. Fig. 3 compares the velocity distribution of the longitudinal section of the bottle cap while Fig. 4 compares the velocity distribution of the transverse section of the

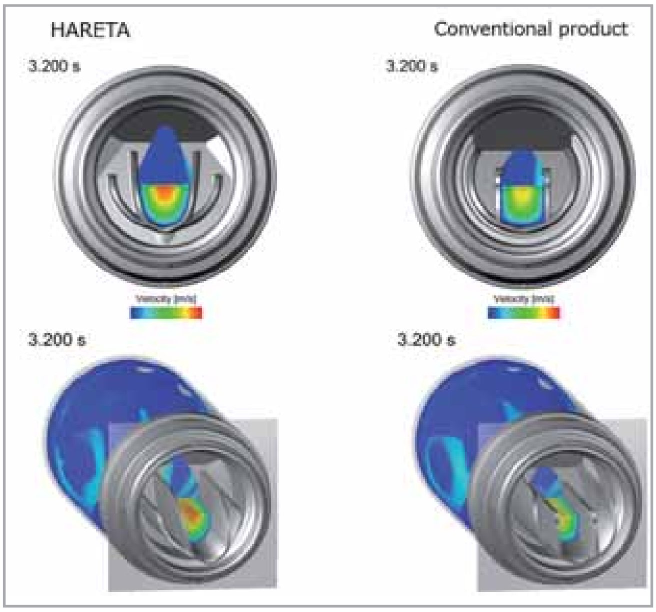
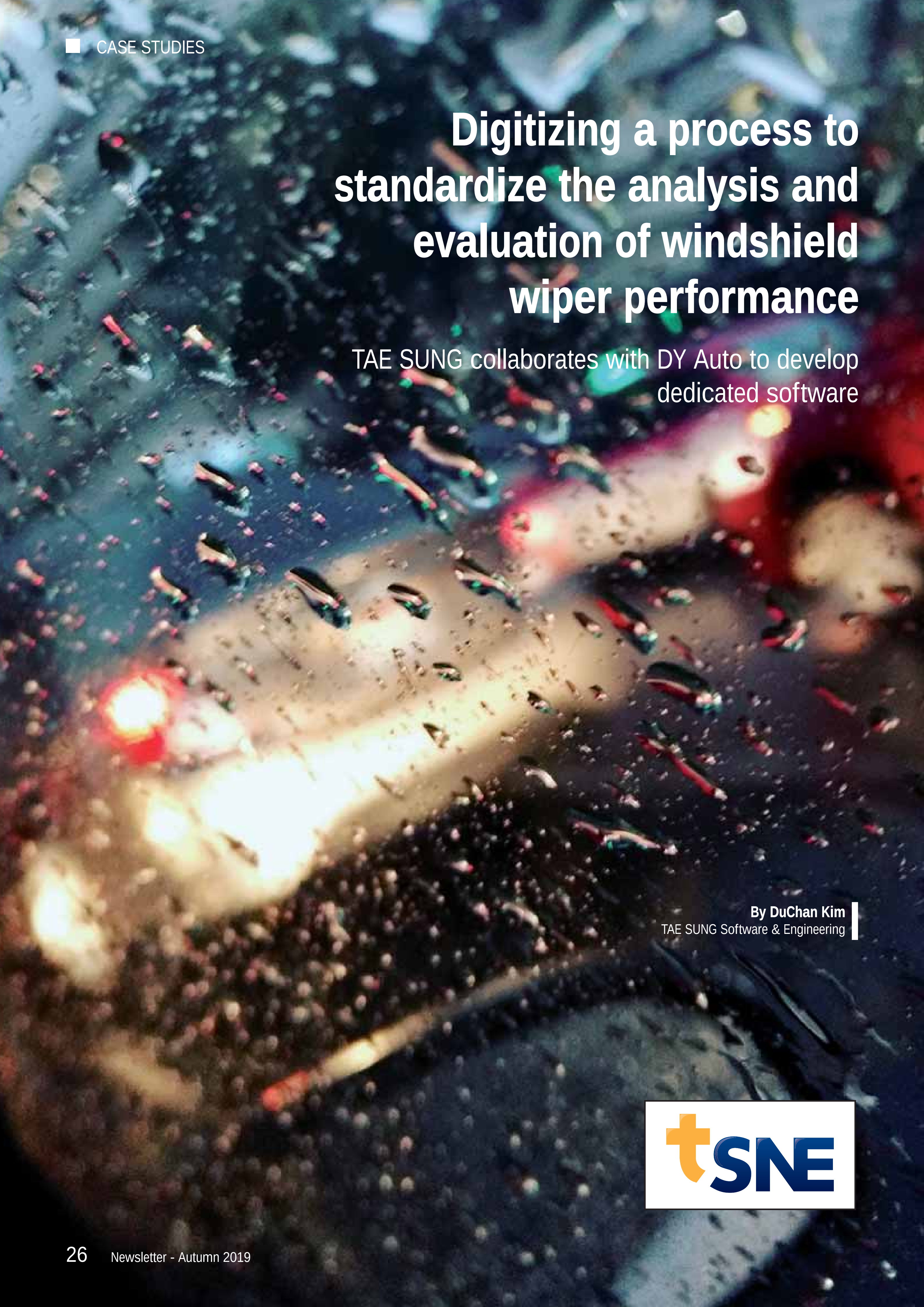


Fig. 4 - Comparison of outflow velocity distribution of detergent in a transverse section of the HARETA spout cap and the conventional product spout cap

bottle cap. This demonstrates that the HARETA cap design has a larger area with high flow velocity. Moreover, experiments on the flow-down velocity using the combination of the newly developed cap for HARETA showed that the HARETA composition was improved by 140%.

Particleworks allowed the CAE engineers to calculate the velocity gradient and the simulation made it possible to observe how much it affected the whole flow-down phenomenon. When the detergent flows down, its velocity is not uniform, slowing near the spout and increasing further away. In future, the engineers hope to use simulation to visualize the differences in velocity gradients due to design changes, feed back the simulation results to the design values, and study the optimal design. In addition, they believe that they can use simulation as a tool for design inspiration because such understanding would often be the origin of new ideas in the design process. The development of this new spout cap was selected for excellence at the Japan Packaging Technology Research Conference in 2018.

For more information about Particleworks
Massimo Galbiati - EnginSoft
m.galbiati@enginsoft.com



Digitizing a process to standardize the analysis and evaluation of windshield wiper performance

TAE SUNG collaborates with DY Auto to develop dedicated software

By DuChan Kim
TAE SUNG Software & Engineering

Fig. 1 - Severe weather condition and various wiper types

to evaluate the performance of the three types of wipers. Since a post-processor is also required to verify wiper quality assessment, the project also developed, for the convenience of users, a pre-processor and a post-processor within a program to build analytical models and analyze the results. The project is updated annually.

The project's main objective was to establish a process to conduct three-dimensional real-scale analysis of any type of wiper, and to establish a specific standard for evaluating wiper performance.

This technical article describes a project that was undertaken by TAE SUNG Software and Engineering, in collaboration with DY AUTO in Korea, to establish a computerized process to conduct three-dimensional real-scale analysis of any of the three types of windshield wiper currently in use, and to establish a specific standard for evaluating wiper performance. It describes the procedure that was followed to model wiper performance, to collect data about that performance in terms of friction, wiping speed and contact pressure, and to develop a user-friendly pre- and post-processor for the developed program to facilitate the creation of analytical models to evaluate wiper performance and to analyze the results.

The purpose of a vehicle wiper is to clear the windshield of the car to ensure visibility for the driver during precipitation. The main variables to consider for vehicle wiper design are to ensure driver visibility, remove foreign substances such as water, snow, dust, etc. from the windshield, and prevent windshield damage. While various ideas have been proposed to substitute them, the existing wiper systems currently remain in use.

There are three types of windshield wiper currently in use: the conventional type, consisting of a simple shaped part and joint; the hybrid type with an integrated yoke and cover; and the flat type which eliminates the yoke and adheres to the windshield using vertebra. Fig.1 shows the three types of wiper and the driver's view during precipitation.

This project was carried out in collaboration with DY AUTO (http://auto.dy.co.kr) in Korea. Rather than conducting the performance assessment using traditional empirical methods, the project required results related to friction, wiping speed and contact pressure (chattering, incomplete wiping, etc.). An analytical process was established that can be generally used

Analysis model

The analysis model was simplified using ANSYS LS-DYNA. The original CAD and analysis models (see Fig.2 and Fig.3) are shown below.

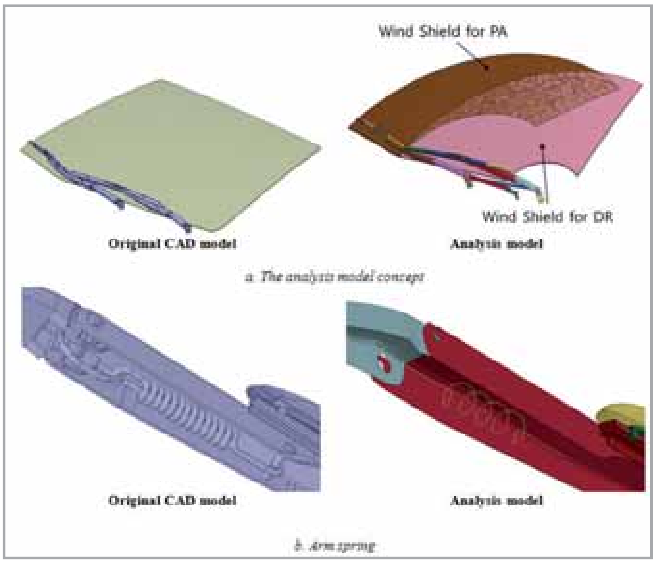


Fig. 2 - Simplified analysis models created in ANSYS LS-DYNA

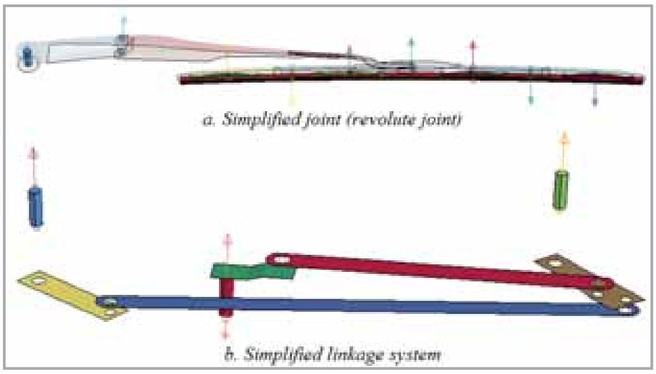


Fig. 3 - Assembled primary parts and linkage system

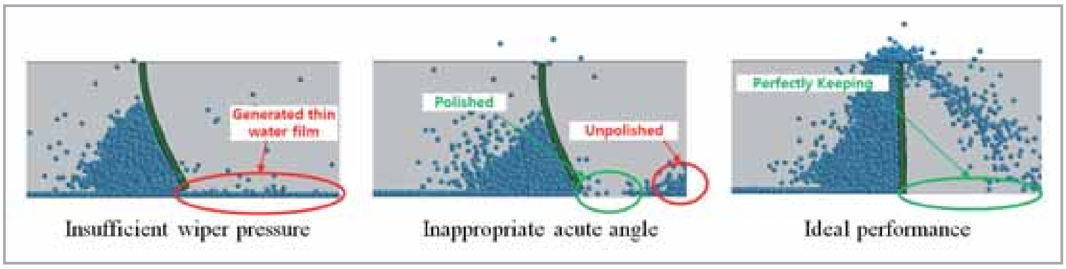


Fig. 4 - Wiper performance based on windshield condition

A wiper generally has three speed levels and it uses a variable motor to maintain the specific speed even if there are substances (e.g. water, snow, dust, etc.) on the windshield. This condition was modelled using PIDCTL in ANSYS LS-DYNA. Traditionally, wiper performance was evaluated with the naked eye based on the condition of the windshield after the use of the wiper. Occasionally, a film of water or water collection patterns may appear (see Fig.4.).

Empirically, the major determinants of wiper performance are the achievement of the appropriate angle between the tip of the blade and the windshield, and the pressure distribution of the blade on the windshield, but it is difficult to obtain quantitative data using experimental analysis. Fixed element method (FEM) simulation represents a simple solution for quantitative data analysis in this case. The flow chart of the project is illustrated below (Fig.5).

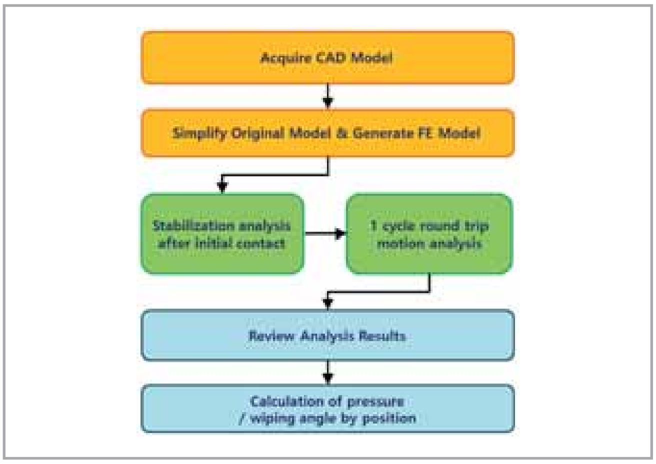


Fig. 5 - Analysis flow chart

Analysis results

Wiper blade pressure can be calculated from the positional contact pressure between the tip of the blade and the windshield. However, no method exists in analysis software to make quantitative calculations of pressure for an acute angle at this

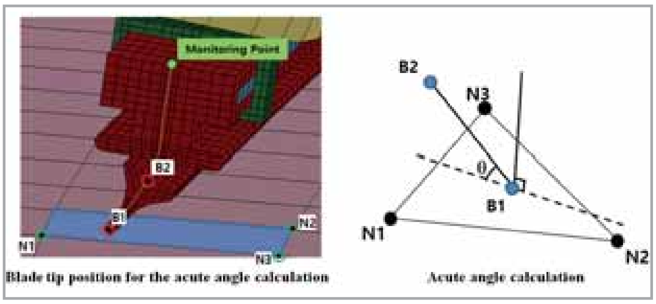


Fig. 6 - Concept for the calculation of blade pressure at an acute angle

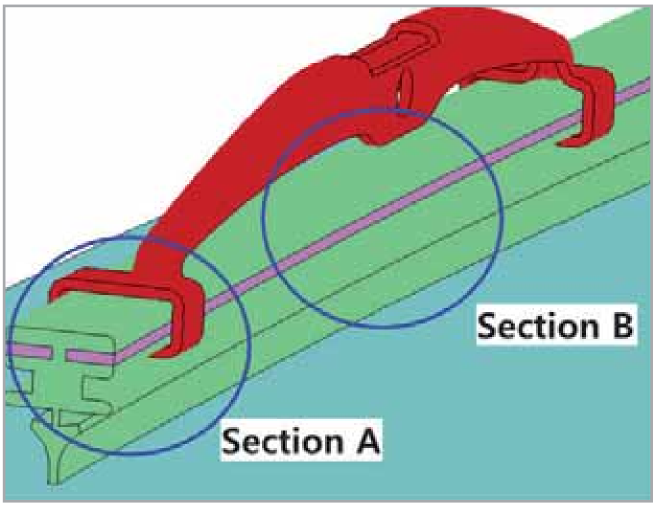


Fig. 7a - Section A: Wiper blade section with yoke / Section B: Wiper blade section without yoke

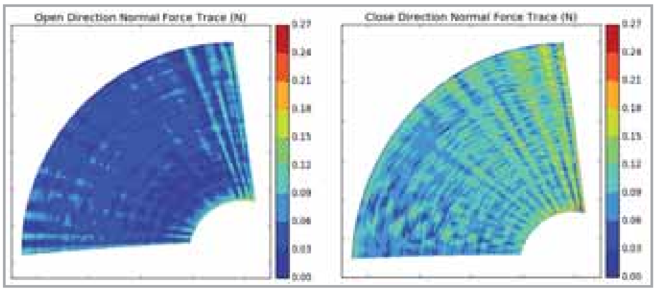


Fig. 8a - Normal force (left: opening direction; right: closing direction)

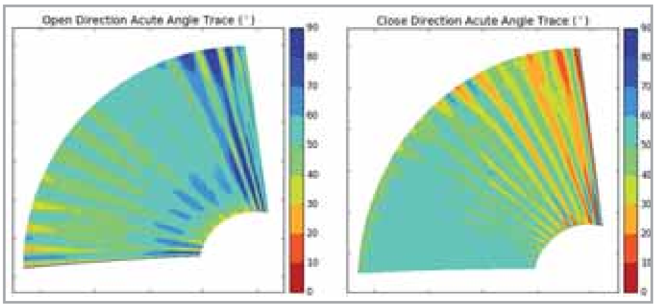


Fig. 8b - Acute angle force (left: opening direction; right: closing direction)

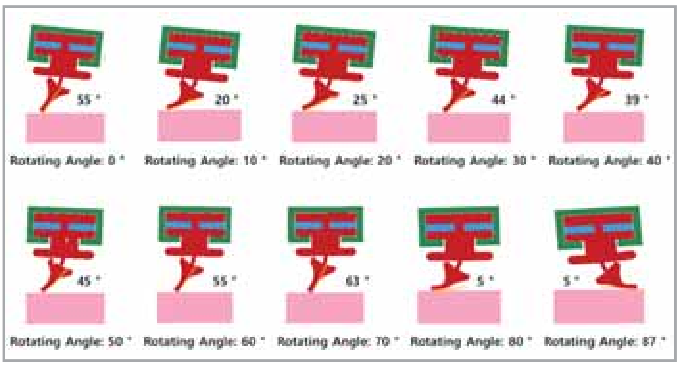


Fig. 7b - Acute angles calculated from wiping angle (Section A)

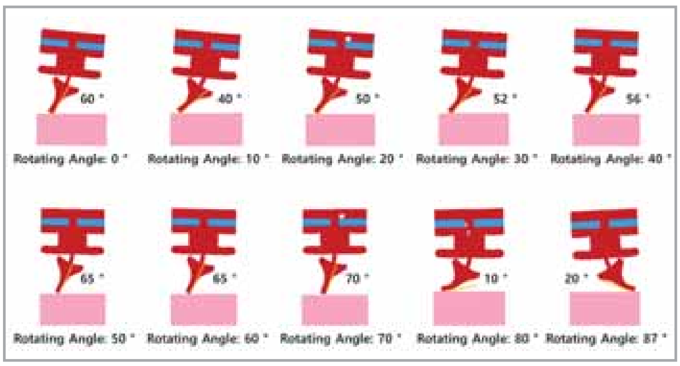


Fig. 7c - Acute angles calculated from wiping angle (Section B) position. To solve this problem, a separate post-processing code was developed using the following concept (see Fig.6.):

- Step1: For each longitudinal section of the blade, designate the top node, which moves with the blade, as the observation point.
- Step2: At every timestep, the program finds the three nodes of the windshield closest to the blade observation point and uses them to construct a plane equation.
- Step3: The acute angle is determined using the angle between the normal of the plane equation and the straight line formed by the two points (B1-B2) on the central node of the blade.

The above steps 1-3 have to be repeated for all sections of the blade over time, a process that was fully automated in the program that was developed.
To verify the success of the program calculations, direct measurements were taken of two of the wiper blade sections (with and without the yoke – see Fig.7) and the results were compared to the program's; the same results was obtained.

The developed program enables the calculation of the pressure distribution and the acute angle based on the wiping angle. The results generated by the developed program for the distribution of the contact pressure and the acute angle by location are shown below (Fig.8).

Fig. 11a - User interface of the pre-processing program developed

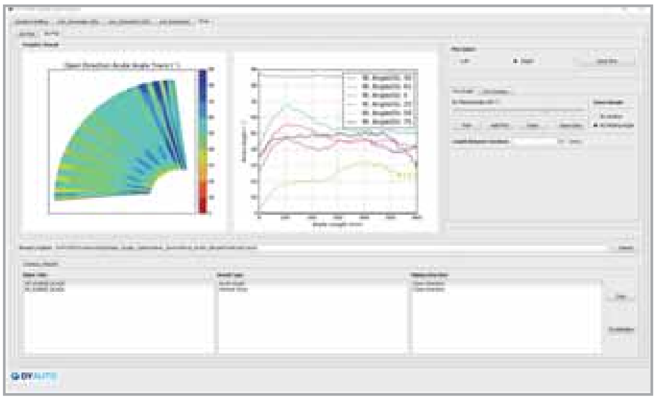


Fig. 11b - User interface of the post-processing program developed

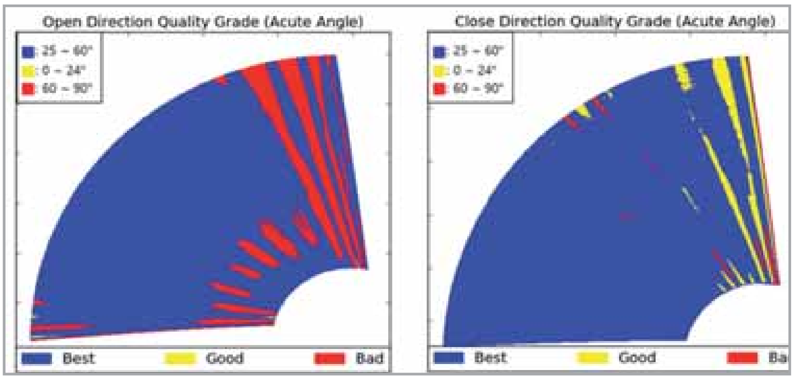


Fig. 9 - Grade of wiper performance quality (left: opening direction right: closing direction)

Wiper performance is optimal when the blade satisfies both the pressure and the acute angle conditions. Therefore, the final performance quality can be calculated by combining the output results (See Fig.9).

Conclusion

In addition to evaluating the performance of wipers, the project also confirmed the friction, contact pressure and the mechanical properties of wiper action. Traditional wiper performance assessments were based on empirical evidence but can also be evaluated quantitatively using simulation, as demonstrated (see Fig.10).

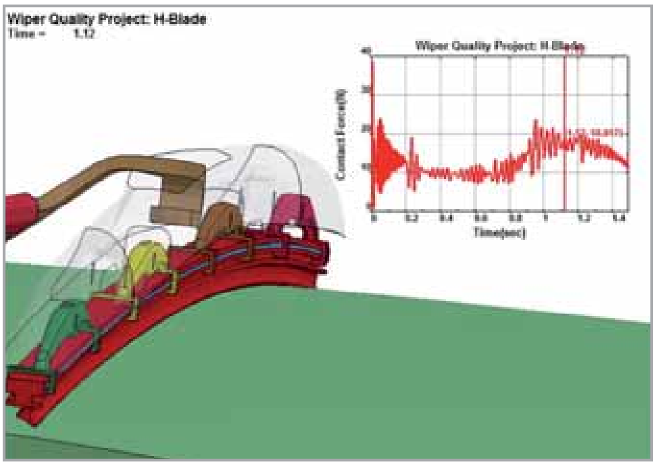


Fig. 10 - Full 3D analysis for wiper quality assessment

We also developed a pre/post-processing program for the project (see Fig.11) to make the program easier to use.

For more information
DuChan Kim - TAE SUNG Software & Engineering
dckim@tsne.co.kr

By Petr Cerveny[1] and Pietro Bianchi[2]
1. Ricardo Software - 2. Leonardo Engineers for Integration

Virtual calibration is growing in complexity, but also capability

Ricardo Software offers a holistic approach that incorporates self-learning

Emission legislation over the short- and medium-term will create very challenging requirements for the development of new powertrains. Increased complexity due to vehicle customization, different hybridization architectures and the market requirement for a vast product portfolio, has generated an astonishing explosion of variants to be calibrated and optimized.

Simulation capabilities are helping test engineers by making it possible to anticipate an ever-widening range of activities through the use of optimization loops using real-time simulators. Real Time simulators need to include human beings into the analysis loops using realistic simulators so that human reactions to vehicle, road and powertrain responses can also be considered. All these techniques are grouped under the term "virtual calibration". This article provides some insight into these methodologies.

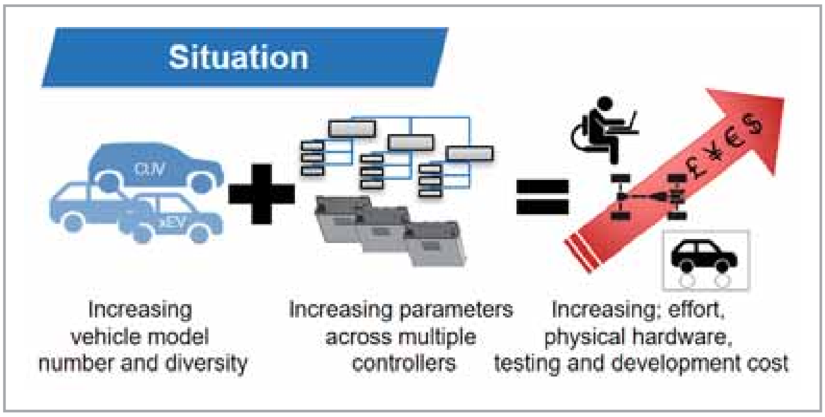


Fig. 1 - Growing product complexity

The indispensability of virtual calibration

Emissions legislation

The limitations of the incoming legislation for most of the vehicles currently in development represent a significant challenge for the planning of calibration activities. Many vehicle configurations require most of the engine's operating parameter settings to be optimized, multiplying the test versions.

Product range complexity

The need to meet the requirements of multiple and often competing customers leads to the extension of product ranges with different engine ratings, transmissions and performance expectations, while the interaction with multi-component platforms creates the need to test, as far as possible, the interaction of many types of controller logic.

The legal requirements for CO_2 reduction and thus fuel economy add further challenges to the calibration task.

Virtual calibration technology

The automatization of what has always been considered a human activity, often based on experience and trial-and-error approaches, can no longer be delayed. The basic building blocks of a virtual calibration technique are:

- Real-Time models of vehicles and engines;
- Optimization software;
- Real-time hardware: depending on the experimental setup and the hardware used, one can consider software in the loop (SiL), hardware in the loop (HiL) or model in the loop (MiL).

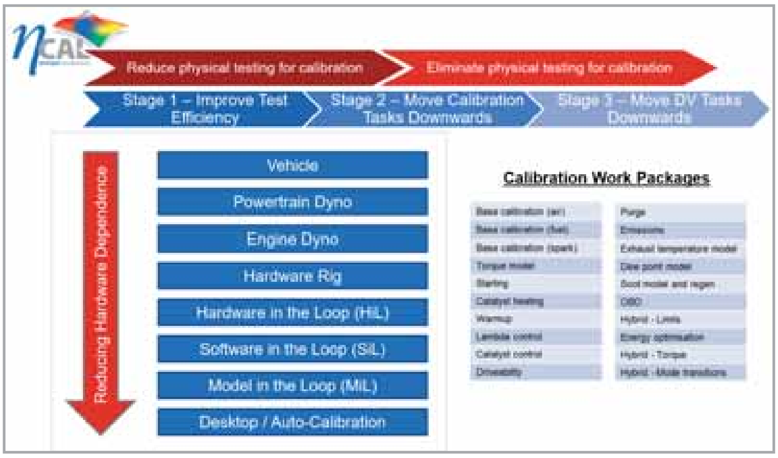


Fig. 2 - Virtual Calibration roles and impact on development

Automatic calibration techniques

Real-time simulation

Real-time simulation is essential to verify the hardware compatibility and that the response times of the physical systems are the same. In the case of a combustion engine, the simulation signals must be supplied to the hardware control at the same, or higher, speed than they are in the real system. From the software development perspective, this is the most challenging aspect: engine performance must be equipped with angular resolution and the reaction to of the actuator statuses must be as fast as with a real engine, without compromising neither accuracy nor fidelity. In particular wave propagation in ducts must be preserved as well as the propagation of the concentrations of the species and temperatures of the fluid must be preserved.

Normally this is derived from a validated "Donor Model", which is transferred into a realtime transient model that produces the same results with reasonable accuracy. "Donor Model" means a virtual model that runs slower than real time, whose fidelity has been experimentally demonstrated on a basic engine configuration before optimization begins. In this case it is possible to reach a further level of efficiency with a direct validation and possible fine-tuning of the real-time model: this is particularly advantageous if the model is directly coupled to the test bench management software, as was recently developed by Ricardo Software with a European OEM.

Real-time solvers are dedicated solvers which prioritize the use of mathematical functions that are performed directly by the processor or are compiled into elementary processor operations. In the case of Ricardo Wave RT, some important accelerations are also provided by the use of polynomial approximations instead of look-up maps. To speed up the solver, outputs are never recorded on disks or other storage media but are made available as signals on volatile memory and accessible via calls to specific public variables, thus simulating their availability on recorded ports as a generic I/O signal. In this way a virtual engine is generated, as an example.

HiL, SiL and MiL

Depending on the type of simulation architecture, we normally distinguish between three different situations:

- Software in the loop (SiL), in which the controller software is tested on a development platform, to test the controller logic and the dynamics of the controllers before accessing the target hardware.
- Hardware in the Loop (HiL), in which the physical CPU with its own software is connected to a real-time PC that simulates an engine: this type of test is usually run to test the complete control system with a virtual engine, in order to evaluate the dynamic behavior of the controller functions, while also taking into account the natural delay of signals. In Spark Ignited engines, for instance, it is of fundamental importance to consider the transport delay of the title and the temperature between the accelerator and the lambda sensor.
- Finally, Model in the Loop (MiL) is used if any part of the controller is based on a mathematical model to assess the behavior of the controlled object or any part of it. In this case the objective of the study is normally the ability of the virtual model to properly support all the functions of the controller. In virtual calibration, the "Model" is a real-time operative model of the engine, which is continuously improved with the results recorded by the test bench.

Software and hardware

Wave Real Time

Since 2007, Ricardo Software has developed a specific solver, Wave Real Time, to simulate both in real time and with a thermodynamic engine with a one-degree resolution. It has been

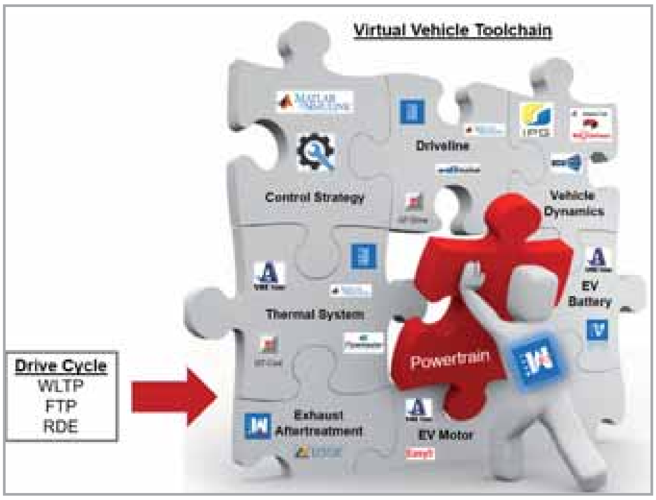


Fig. 3 - WAVE RT Role in Virtual Calibration

extensively used by major North American and European OEMs for over a decade for a wide spectrum of applications, ranging from software definition, to hardware testing and, finally, to virtual calibration. It outputs a C code that can be compiled on any real-time PC such as ETAS, dSpace and National Instruments hardware. An application has also been create for Raspberry PI modules.

Wave RT models can be produced by a Wave donor model with an embedded translator, which generates object code in minutes, and which can be compiled with proprietary libraries on target machines or included in an S-Function to be compiled within a MATLAB executable.

Unlike the old fast running models, donor models do not need to be simplified or modified to be translated into a real-time one, as the latter is equipped with a specifically developed solver based on a modified Method of Characteristics, which allows for the propagation of state variables along the ducts, thus naturally including all delays due to transport phenomena. Thanks to its functional mockup interface (FMI), Wave RT models can be imported into vehicle dynamic simulation software, such as IMG CarMaker® or V-Grade® for real-time simulations or for real drive or ride simulators.

ŋCal

ŋCal is a platform used by Ricardo engineers to manage the process of reaching a potential optimum calibration. It manages adjustable parameters, assesses results distinguishing the effect of noise over the relevant results, improves engine models with test bench results, and drives next steps in the optimization process. It is MATLAB-based and uses a knowledge database from past projects.
Its objective is to create a shortcut between the prototype phase and the implementation phase, enabling detailed development to take place in the first period of the project and increasing their value and significance in the subsequent ones.

IGNITE

IGNITE is a software product, developed by Ricardo Software in the last decade, and based on Modelica to exploit its acausal solvers and FMI interfaces. It aims to describe any physical model that can be described with ordinary differential equations. It is mainly dedicated to the simulation of vehicle dynamics, both longitudinally and three-dimensionally.
IGNITE is equipped with a library enriched with multiple vehicle components, from the engine to the tires and can import the Wave RT model for real-time vehicle simulation, making it suitable for parametric optimization at the vehicle level.

Compatibility

As previously stated, compatibility with other simulation platforms is a fundamental feature: Wave can import GT Power ® models and transfer them to Wave Real Time. Such models can also be exported to Simulink ® or to any FMI-compliant platform, including AmeSim® and many others.

Ricardo's approach

Ricardo promotes a holistic approach to virtual calibration through an architecture that includes at least two interacting self-learning loops: the first incorporates the calibration accelerators that use the Real Time model to generate the subsequent steps of the optimization, and the second uses the test results to constantly improve the model of the engine and vehicle, increasing its fidelity and forecasting capabilities.

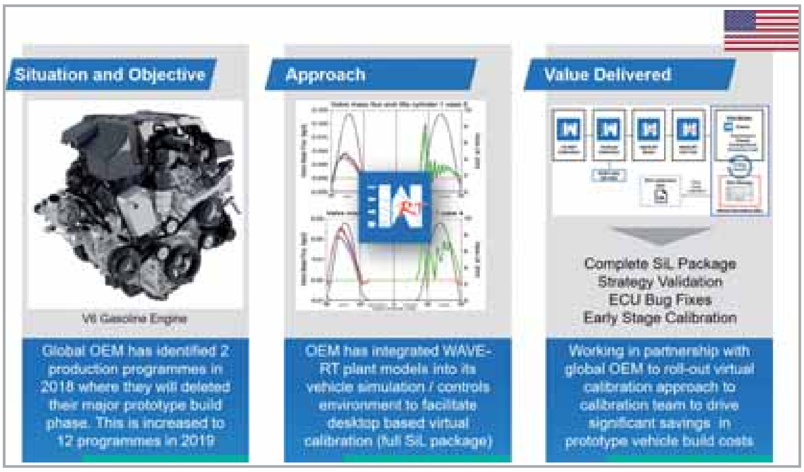


Fig. 4 - Success stories: USA market

Normally the real-time model of the engine includes all the ducts and tubes up to the catalyst, where a pre-processed kinetic chemistry simulator generates conversion efficiency figures based on the thermodynamic state variables at the catalyst inlet. The vehicle is normally modelled with its full dynamic powertrain, including the manual or automatic transmission, electrical transmissions and batteries, if any, and includes the possibility of modeling different driving styles with a sophisticated driver model.

ŋCal provides guidance on progress and the continuous improvement of performance and processes parameters, results and target values.
Some examples of past success stories in the USA and Europe follow.

Success stories

USA

Virtual calibration was introduced in two pilot projects for the production of gasoline engines. An architecture was implemented in which parallel engine tests and real-time engine simulation were carried out. The engine model was paired with a complete Simulink® vehicle model to simulate the World Harmonized Transient Cycle (WHTC) and related emissions. This model was able to progress to the subsequent steps of the optimization path, while the results of the test bench at the new points were used

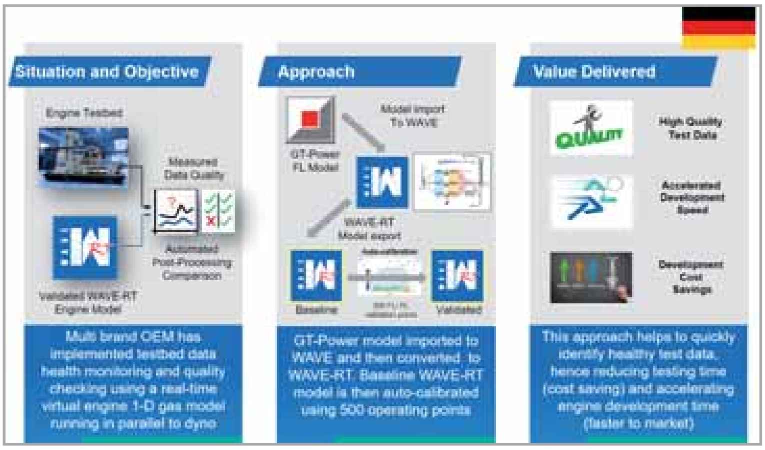


Fig. 5 - Success stories German market

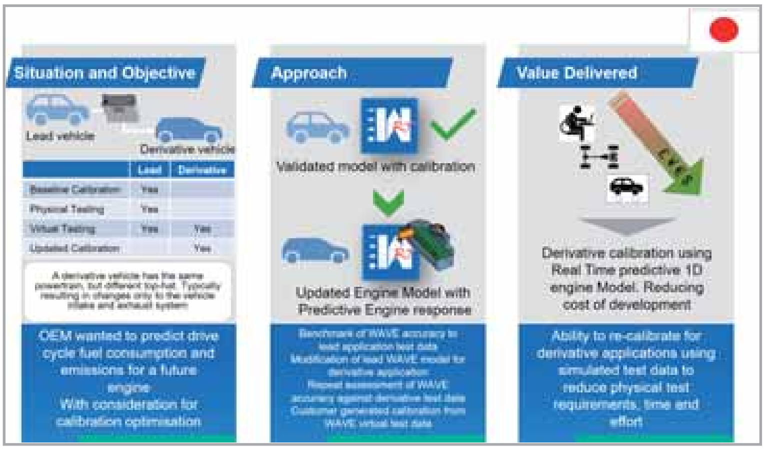


Fig. 6 - Sussess stories: Japan market

to fine-tune the engine model with increasing precision. The customer was able to eliminate at least one batch of development prototypes, while the methodology was extended to a further 12 production projects currently under development.

Germany

Some developments with a German customer were particularly interesting because they allowed the transfer of the combustion test data collected from a single-cylinder test rig to a multi-cylinder model, which was used to extend the combustion improvements to the multi-cylinder engine.

Using a predictive and tunable combustion model available in both Wave and Wave RT, the results of the single-cylinder test rig were used to generate a suitable donor model of the multi-cylinder engine, which could be used for its virtual calibration.

The value for the OEM was clearly the possibility to take advantage of the information available in the alpha phase of the project, effectively reducing the experimental effort required during the beta phase, performed with multi-cylinder prototypes.
Some other projects aimed to develop and calibrate the logic of onboard diagnostics (OBD), with the possibility of testing them on the Real Time Simulator, which include the drivers' psychological reactions in the experimental perimeter.

Japan

Many projects using virtual calibration techniques have been conducted in Japan. Some included projects to assess the cold part of driving test cycles, deriving the setup for cold combustion from a single-cylinder test rig and simulating the complete cooling system during a cycle to take the heating dynamics into account.
This process enabled the generation of calibration strategies for a cold start months before any vehicle prototype could be produced, reducing the need for costly track and road tests and increasing the success rate of the early tests.

Conclusions

Upcoming stringent emissions legislation has introduced new requirements for sophisticated simulations. Compliance with cycle-based emission limits requires highly efficient and reliable analysis that can predict total gaseous emissions and fuel efficiency over several cycles. Hybridization introduces further complexity into the issue, requiring the simulation of complex recharging, service and regeneration logics in time scales of a fraction of a second. Human interface and human reactions also play a role in determining the final results, so real-time simulators now include a human being in their test loops.

Ricardo Software has been developing real-time thermal engine cycle simulation since 2006 and launched Wave Real Time in 2008 for the first time. Since then, all major OEMs around the world and Formula 1 teams have harnessed its capabilities so that a variety of applications are now possible, while dozens of application experts have matured sufficient knowledge over the past decade to be able to address all the most demanding engineering challenges.

Virtual calibration has, therefore, become an achievable, albeit challenging, goal for most OEMs, and both the availability of Real Time software and the experience accumulated over the last decade have enabled very efficient, robust and reliable systems to be implemented worldwide with the widest variety of applications.

For more information
Pietro Bianchi - Leonardo Engineers for Integration
pietro.bianchi@leonardointegration.com
or visit: www.ricardo.com

The future of vehicle systems engineering

Using a single environment for truly acausal physical modelling

by Alessandro Picarelli, Mahdieh Mehrabi
Claytex

The overall complexity of vehicles of all types is increasing along with customer expectations for efficiency, comfort and reliability. This means that the multiple systems in the vehicle, which span multiple engineering domains, must be optimized to work together efficiently. Physical systems modelling that extensively integrates the different physical domains (mechanics, fluids, electrical, control, etc.) is essential. If the models are sufficiently accurate, physical systems modelling can significantly reduce the number of prototype builds, reducing development time.

This article explores the additional and significant benefits of using an acausal modelling language to model the entire plant in a single environment.

Electric vehicles (EVs) are gaining popularity with several new vehicles being launched this year. In many cases, these vehicles are mechanically simpler than conventional and hybrid vehicles, but they still pose many systems integration challenges for the original equipment manufacturers (OEMs).

In all vehicle types, the overall level of complexity is increasing along with customer expectations for efficiency, comfort and reliability. To meet these challenges, multiple systems in the vehicle must be optimized to work together efficiently, and these systems span multiple engineering domains. For instance, energy usage of the vehicle must be expediently managed whilst also maintaining occupant comfort levels and ensuring that every vehicle system operates safely and efficiently. Often, individual subsystems have to be compromised to achieve the overall optimum system performance. Therefore, developing the subsystems in isolation is no longer an option.

Physical systems modelling with extensive integration of different physical domains (mechanics, fluids, electrical, control, etc.) is essential – both to guarantee the achievement of attribute targets and to support the testing of the control and diagnostic systems. Assuming the model fidelity and validity is sufficient for the aforementioned tasks, one can achieve a significant reduction in the number of prototype builds and in development time.

Methods of achieving systems integration

Sequential simulation: traditionally over a duty cycle you will generate, for example, heat losses during the operation of a particular drive component. These results will then be used as boundary conditions for a cooling system and, as a result, the cooling system controller will be developed. The problem with this type of simulation is, as the name suggests, that it is sequential and therefore will not model an action-reaction behavior. Changes in the cooling system will not affect the drive component's operation.

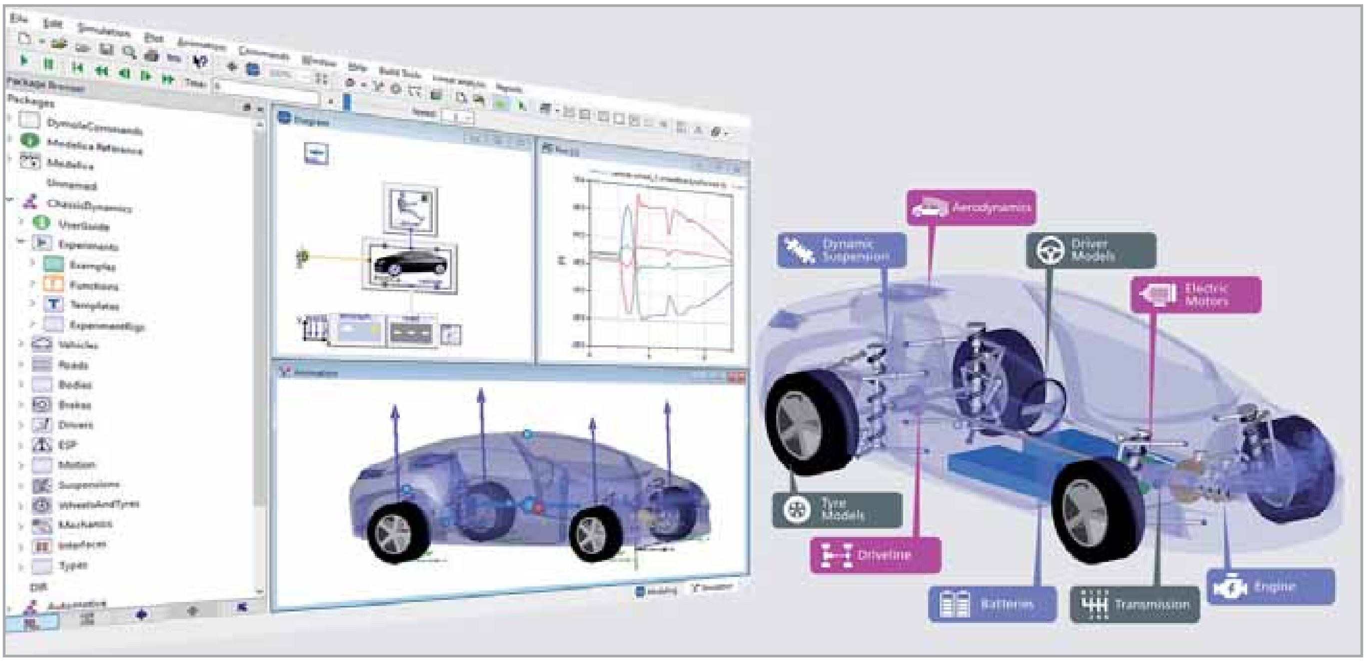
Co-simulation: partly addresses the issues of sequential simulation and the inefficiencies of interfacing two or more physical models created in different specialized environments. The communication step size must be chosen carefully to ensure that the information is exchanged at a sufficiently high rate to yield accurate and representative results. Whilst interfaces such as Functional Mock-up Interface (FMI) support this type of modelling, the suggested approach is to connect one plant model to one or more control models.

Multi-domain simulation: enables the user to model the entire plant in one environment, exploiting the capabilities of model code optimization, or symbolic manipulation (in the case of acausal modelling languages such as Modelica), where the user is not required to determine the execution order of the equations for the physical model.

Multi-domain simulation can offer huge benefits in modelling effort, particularly when using acausal modelling languages where the same component model can be reused in any possible scenario, including model inversion, which is otherwise impossible with procedural code. Unfortunately, the benefits of acausal modelling have to be experienced directly by engineers to be fully appreciated.

Multi-domain acausal vs causal models

There are several points that systems engineers must understand when considering acausal (Modelica-based) vs causal (procedural code-based) solutions. Acausal models describe how the parameters (variables and signal) and physical connectors are related to each other through equations. These equations are set out in the code in an arbitrary arrangement where the order is not important. When the Modelica code is compiled, for instance in DYMOLA, all the system component equations are extracted and rearranged to find the optimal calculation order in relation to the knowns and unknowns, which are automatically identified within our model. Any changes in boundary conditions, for example from flows to pressures, or temperatures to heat flows, are dealt with efficiently and automatically and no manual rearrangement or reformulation of the equations is required. This would be impossible without arbitrary



DYMOLA environment example with seamless multi-domain integration of systems declarations of the system behavior using physical relationship-based equations; the modeler would be responsible for taking up this tedious and error-prone task.

Example application

In vehicle thermal modelling, we have helped customers evaluate the benefits of new glazing and panel insulation technologies using multi-domain acausal approaches, not only to predict the thermal comfort of the occupants but also the resulting loads on the heating, ventilation and air conditioning (HVAC) system, the compressor drives (electrical or internal combustion engine-based), and the effect of these on the engine controllers and battery management system (BMS) as well as on tailpipe emissions (where these apply), not to mention on the integration of the powertrain cooling systems and the aerodynamics. For these types of models, the parameter information is usually readily available from the glazing manufacturers, and the software allows the user to specify layered glazing and panels within the vehicle including air and other gasses, where applicable.

The cost benefit of test virtualization will depend on the number of tests that the vehicle developer was planning on doing during development, the labor rates, and the facilities to hand. For example, if the instrumenting and testing is outsourced, the virtualization cost benefit will be higher. Instead, if the preparation of the vehicle is done virtually, it will inevitably involve fewer people and, therefore, potentially fewer sources of error. These errors in vehicle preparation planning and implementation can, in some cases, nullify the entire test.

The "FMI will remove the need to consolidate software tools within my company" illusion

In the HVAC and overall vehicle energy efficiency example above, all the systems and subsystems involved are simulated within one systems engineering environment, DYMOLA, thus avoiding the pains of co-simulation between a range of tools. Based on our experience, the only co-simulations we would recommend would be CFD-1D/0D, and with controllers and controller models.

Acausal models, particularly fluids, are best simulated within a single environment. Furthermore, when using acausal modelling languages such as Modelica, engineers appreciate the ability to exploit the advantageous symbolic manipulation algorithms throughout the entire physical model. This means code optimization can be applied to a greater part of the physical plant model, leveraging the benefits that this delivers in terms of CPU time.
We had been preaching this for a while and, in 2015 at the 11th Modelica Conference in Versailles, the speaker from a major vehicle OEM openly stated in their presentation that the company had come to realize that FMI could not replace their software tool rationalization task, as they had initially thought. Indeed, interfacing such a broad range of software tools (hundreds) proves almost impossible for a variety of reasons, such as the communication overheads, and the choice of adequate communication step sizes to yield the correct results, among many others.

However, there are still users who hope that FMI and similar technologies will seamlessly resolve all their interfacing problems for models generated using different tools throughout a company's many engineering departments. Instead, it should be used in those cases where plant models are interfacing with controllers, or potentially within the plant model, when interfacing system level models with computational fluid dynamics (CFD)/finite element analysis (FEA)-type models, where the system models provide the boundary conditions for the CFD/FEA.

At the 2017 Automotive Thermal Management Forum in Manchester, the main OEMs in attendance confirmed our strong, established belief that the solution to avoid the heavy use of FMI was to use a single modelling environment to model as much of the plant and as many of the controllers as possible. This single modelling environment had to be suited to physical modelling and would preferably offer a truly acausal physical modelling of the plant.
At Claytex, we have been using Modelica and DYMOLA for the past 21 years and continue to do so because of the nature of the language and of the range of functionality within the language that DYMOLA enables you to use. Whether we are looking at the performance of the condenser and evaporator within an HVAC or heat pump system, or indeed the performance of the entire vehicle multibody chassis and suspension, we can analyze it all within the same environment.
The symbolic manipulation algorithm within DYMOLA allows the models to run fast enough for a range of applications, from detailed vehicle modelling within driver-in-the-loop environments through to advanced driver assistance systems (ADAS) and connected and autonomous vehicle (CAV) testing, all of which require excellent representations of vehicle models to ensure that the results accurately reflect real-world performance. The physics-based approach allows models to be predictive, rather than solely empirical, although empirical models can also be included.

The philosophy behind the development of the Modelica library involves no "black box" modelling (although encryption is possible when required to protect intellectual property). The fact that most of the commercial libraries have code that is visible to the Modelica users is advantageous in many ways. Firstly, the users can see for themselves exactly which equations have been implemented in the models to represent their behavior. Secondly, users can build upon the existing code or even customize it to suit their applications, which is particularly useful when developing bespoke or new technology that is not included in the libraries. The drag-and-drop approach to building models still allows users to see what is going on behind the scenes and to edit this, if required.

As vehicles and systems in general become increasingly complex, interconnected and, by nature, multi-domain, we need tools and expertise to model the physics as efficiently as we possibly can to deliver robust products. DYMOLA and Modelica enable our customers to do this.

For more information
Alessandro Picarelli - Claytex
alessandro.picarelli@claytex.com

Machine Learning in Engineering

Use of machine learning for engineering applications brings field data and engineering knowledge together, resulting in more accurate decision making.

By Dr. Fatma Kocer-Poyraz
Altair

When we think of machine learning applications, the first things that come to mind are everyday life examples such as spam filtering, various recommendation engines and fraud detection systems. These are direct applications of machine learning that simplify routine tasks, making our lives more productive, enjoyable or safe. Less well-known are applications of machine learning in engineering that are important in improving our productivity and safety.

Machine learning was first defined by computer scientist Arthur Samuel in late 1950s as "a field of study that gives computers the ability to learn without being explicitly programmed." It differs from traditional programming in several ways.

In traditional programming, inputs are system characteristics and a program of rules that govern that system; the output is the system's performance. In machine learning, inputs are data that includes both the system characteristics and performance for many system variations, and output is a program that can predict the system output based on the system characteristics. As a result, in traditional programming the program grows by fine-tuning the rules while in machine learning the program fine-tunes with additional data. In the scope of Computer Aided Engineering (CAE), traditional programs are referred to as physics-based simulations and machine learning models as data-based predictive models.

Similar to how physics-based simulation programs are categorized with respect to the physics they are solving for, machine learning algorithms are also categorized with respect to objectives and data characteristics. In general, machine learning algorithms are classified as supervised or unsupervised.

When the goal is known, and the data has labels, that is considered supervised learning. Examples of this include predict- ing house prices based on house sizes, part stress based on part dimensions, or bearing status (normal or irregular) based on accelerometer data. In unsupervised learning, the goal is to find patterns in the data, and data does not have labels. Examples of this include predicting credit fraud given credit history or predicting anomalies in bearings given accelerometer data. Supervised learning can be of regression or classification type whereas unsupervised learning is of clustering type.

At Altair, we have been using regression type machine learning algorithms for design exploration and optimization for over 30 years. Our recent acquisition of SmartWorks, our Internet of Things (IoT) platform (formerly Carriots) has expanded the scope of in-house expertise in machine learning. This article addresses engineering applications that are successfully employing the use of machine learning.

Predictive Modeling and Prescriptive Analytics of CAE or Test Data

In the first application, Altair Multi- disciplinary Design Optimization Director (MDOD) uses simulation data for supervised learning. To meet today's demanding requirements for product performance and its time-to-market, the use of Multi- disciplinary Design Optimization (MDO) has become a need.

Those familiar with MDO applications are well aware that setting up and solving MDO problems can be labor intensive and computationally expensive, especially if the application is large-scale such as an automotive Body-in-White (BIW) design. Altair's MDOD is an environment developed to break these barriers for utilizing MDO in engineering applications. Its model-centric environment utilizes automated model linking and a rich visualization for ease of setup and review. It also employs efficient and extensible sampling methods that are conducive to machine learning. For optimization, MDOD leverages efficient machine learning based on global search methods for computational effort reduction.

A case in point highlights the predictive modeling and prescriptive studies that were performed for a fabric softener bottle redesign to sustain new transportation conditions. Unilever Corporation needed to increase the load carrying capacity of a fabric softener bottle without adding to the bottle weight. Altair HyperWorks™ was used to parametrize the design features.

A Design of Experiments (DOE) study was used for feature reduction, from which a predictive model was created. Finally, a prescriptive study was conducted using this model. The resulting design had 20% more load carrying capacity, while at the same time achieving mass savings of 5%.

Merging Physics-Based Simulations with Data-Driven Predictive Models

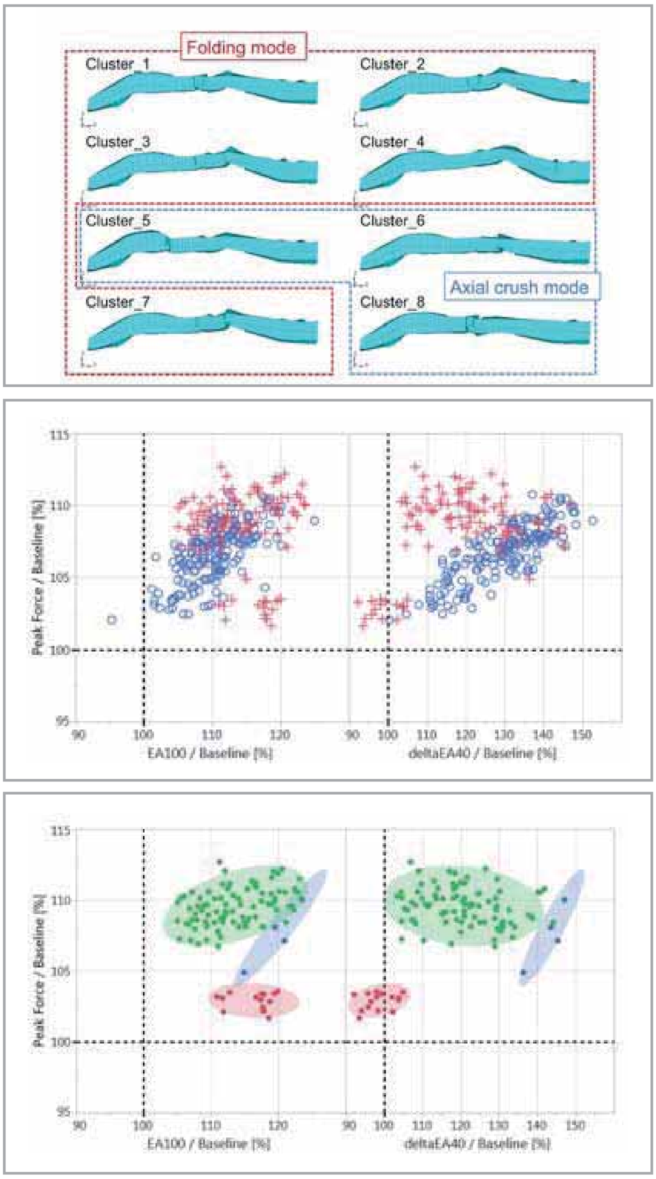
In a research project with one of Altair's major customers, machine learning methods were successfully employed to create a concept design for a reinforcement bracket subjected to crash loads. Concept design methodologies using topology optimization often have difficulties when exploring highly nonlinear events such as crash applications. In this particular application, we had to avoid any folding modes after the crash event.

To be able to include this key performance indicator (KPI) as a response in the optimization study, the design space was sampled first. Clustering was then used to group them with respect to their folding mode. Once the clusters were known, the consecutive designs could be classified. Using regressions and classification, the design was optimized to achieve the desirable folding mode and high- est energy absorption.

Merging Historical and Real-Time Data

Aircraft engines go through a number of cycles of demanding and varying conditions. To avoid catastrophic failures, they require a very thorough maintenance pro- gram. Maintenance of engines is costly due to engine downtime and the time and labor involved.

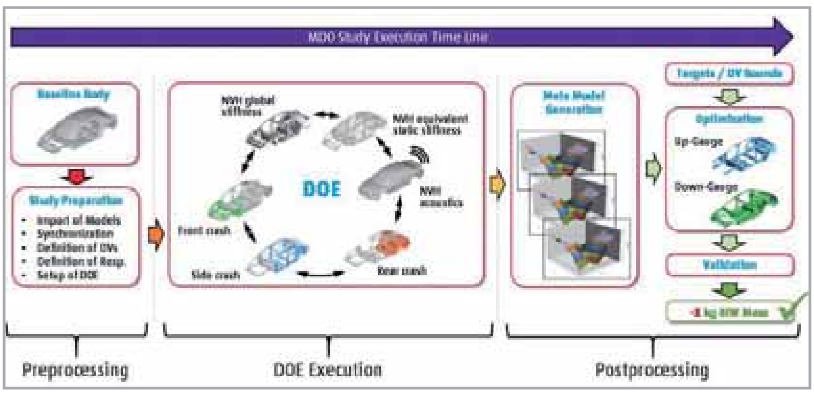
The engines are equipped with a number of sensors measuring temperatures and pressures at various locations as well as other important measures such as fuel- to-air ratio and fan speed. Using machine learning algorithms, this sensor data can be used to make predictions of engine health based on the conditions it has been sub- jected to through its lifecycle.



Altair Multi-disciplinary Design Optimization Director (MDOD) Process

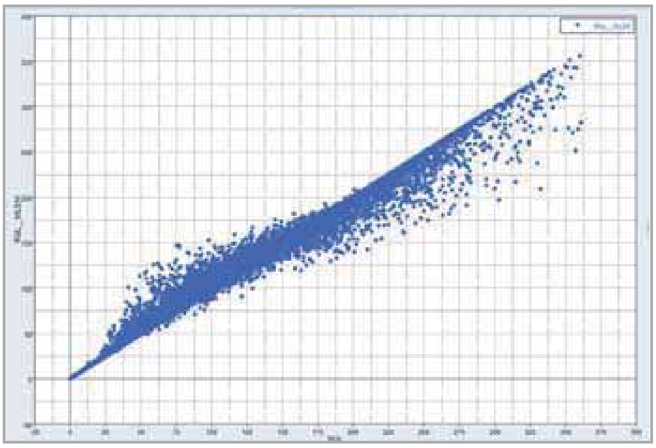
The NASA Prognostics Center of Excellence (PCoE) at Ames Research Center provides two sets of sensor data to be used for model training and testing (https://ti.arc.nasa.gov/tech/dash/groups/pcoe/prognostic-data-repository/). Included in the training data set are 21 sensors' data from 100 engines that ran to failure. In the testing data set, the engines remain in operation. The objective is to predict the Remaining Useful Lifecycles (RUL) before the engines in the test set fail. To solve this problem, the RUL was defined and computed. The data was then reviewed and cleaned. Most importantly, the characteristics of the data such as important sensors were identified. An automated predictive modeling technique was used to come up with the best model based on its cross-validation coefficient of determination, or R2. This model was then employed to predict the RUL of the engines in operation.

As demonstrated by the results in the table, the predictions for engines with sen- sor data for 100+ cycles were on-target. Since the engines on average go to 200 cycles, the predictions become accurate



Altair Multi-disciplinary Design Optimization Director (MDOD) Process

halfway during the total lifecycles. For the engines with less data, with no patterns emerging, the predictions will have larger confidence intervals. An example of an application for unsuper- vised machine learning is related to bearings – a critical component in most industries. PCoE provides bearing accelerometer sensor data for four bearings at a sampling rate of 20kHz, resulting in sampling every 10 minutes for a duration of 1 second for 9 days. This totals 20 million records. The first sampling corresponds to the new bearing and is used as a reference for anomaly detection of the bearing as it ages.
In this project, the objective is to recognize these anomalies as soon as they occur, before they lead to irreversible issues such as part failures. In the machine learning process, first Principle Component Analysis (PCA) is used for feature reduction, then the samples are correlated to the healthy sample. Finally, the anomaly is detected using a threshold for correlation drop and value.



Actual vs Predicted Engine RUL

So far, a supervised regression problem (engine RUL prediction) and an unsupervised machine learning problem (bearing anomaly detection) have been covered. Lastly, a supervised classification problem example based on an APS Failure at Scania Trucks Data Set from the UCI Machine Learning Repository will be reviewed.

The data is provided by Scania Trucks, but attribute labels have been changed to disguise the data. In this problem the objective was to minimize the maintenance costs of the air pressure systems (APS) in trucks. Maintenance was required to avoid a failure while trucks were in operation. As it is a costly procedure, on the one hand, it was desirable to perform maintenance only when needed. On the other, repair costs associated with a failure would be significantly higher. If the predictive model failed to predict a maintenance (false negative), the truck would fail during operation and it would cost 500 units to repair. If the predictive model showed a need for maintenance but in fact it was not needed (false positive), it would waste 10 units.

This data set has values for 171 attributes used in predicting whether maintenance was needed or not. The training set contained 60,000 rows of which 59,000 belonged to the negative (no maintenance required) class and 1,000 to the positive (maintenance required) class. The test set contained 16,000 examples that needed to be classified as neg- ative or positive.

Engine No.	Current Cycle	Actual RUL (cycles)	Predicted RUL (cycles)
101	31	112	171
102	49	98	139
103	126	69	65
104	106	82	81
124	186	20	19
125	48	145	160
134	395	7	11

Actual vs Predicted RUL for Engines in Operation

The first step in data-based modeling is to review the data for missing values and errors. This particular data had some attributes missing up to 82% of values. This could be due to lack of time to measure all attributes in between operations. This also meant that attributes with such high amounts of missing data would be the ones ignored by employees as not critical. As a result, eight attributes that have more than 50% missing values have been removed.
For other missing values, median imputation was used. With the remaining data, PCA for feature reduction was conducted on the remaining 162 attributes. It was found that 99% of variance in the data was explained by the first 110 principle components. Weights were allocated to classes for class imbalance in the data.

The resulting predictive model cost for this dataset was only 10,710 units, with 671 false positives and 8 false negatives. This quality is one of the best publicly reported in the literature.
Machine learning is not a new concept; it has been around since the 1960s. However, all the elements required for machine learning have come together only recently. Access to more bandwidth to transfer data over the Internet, affordable data storage, and increasingly powerful computational resources is now widely available. Most importantly, universal accessibility to robust algorithms democratizes the use of machine learning. With the growth of IoT and access to field data, the synergy between design and operation can easily be created and leveraged for better products. As engineers, our job is to identify the applications that can benefit from the use of machine learning and develop easy-to-use processes for them. The Altair MDO Director is a good example for this. It simplifies the creation and collection of data from different sources, it creates predictive models using machine learning and uses these to f ind optimal designs for an MDO problem, all done in an environment with an intuitive user interface.

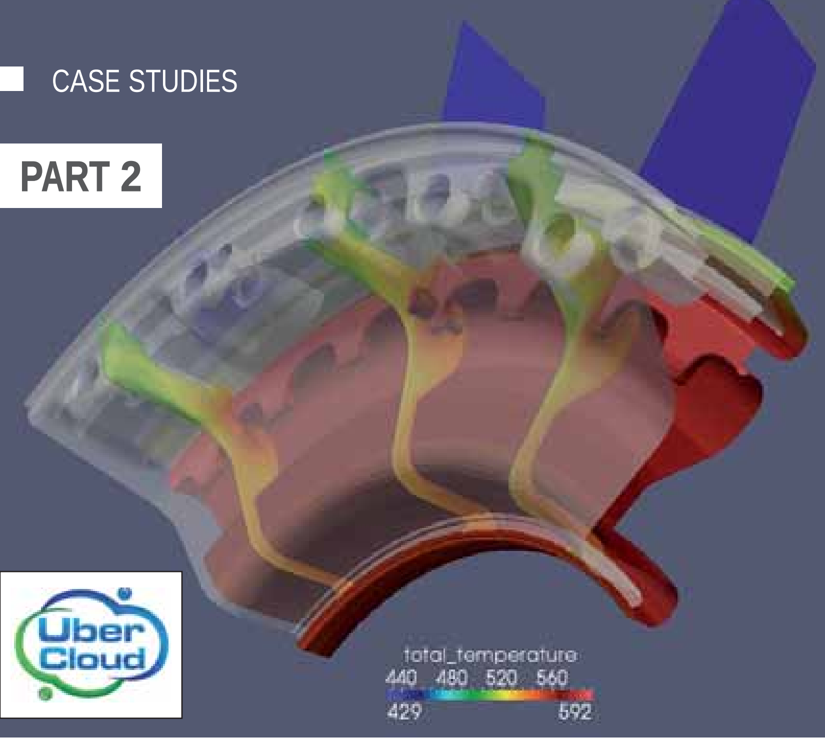
There are many other engineering applications of machine learning. For more information please visit www.altair.com/iot.

Altair is a sponsor of the International CAE Conference & Exhibition 2019 and participates with a presentation in the Manufacturing session.

www.caeconference.com

PART 2

Engineering simulations using ANSYS in the cloud



By Wolfgang Gentzsch
UberCloud

This is the second part of an article presenting a selection of case studies dealing with engineering simulation in the cloud using ANSYS software LS-DYNA, HFSS, and ANSYS Discovery Live; the first part appeared in the previous edition of the EnginSoft magazine and presented case studies based on ANSYS CFX and Fluent [1]. This article will briefly describe three engineering use cases, the cloud implementation, the major benefits and challenges, and will conclude with the lessons learned and recommendations on successfully moving engineering workload to the cloud.

Engineering companies have only recently begun considering using the novel cloud computing model. Major reasons for this hesitation include security concerns, traditional software licensing models that were not ready and flexible enough for the cloud, the size of the data transfers, engineers' concerns of losing control over their assets, and internal resistance from management and IT departments that would often barricade themselves behind 30-year-old compliance regulations.

After seven years of developing and implementing our engineering cloud services, and running more than 220 high-performance computing (HPC) cloud experiments and over 120 engineering case studies (30 of which were based on ANSYS software), UberCloud has been able to remove almost all these roadblocks by developing a software technology that hosts the engineer's complex workflows in a container that can be housed on a dedicated HPC resource in any cloud – public or private, hosted and hybrid.

The ANSYS and other case studies are available for download [2].

An airbag simulation with ANSYS LS-DYNA in the Microsoft Azure Cloud

Automobile airbags were created with some incredible engineering. In a high-speed crash, the driver of the car can be hurled against the steering wheel but, in an airbag-equipped car, an airbag, inflated by a small electronic device, provides enough cushioning to protect the driver during impact. Fatalities and serious injuries have been reduced since the widespread installation of airbags. Undertaken by CAE consultant, Praveen Bhat from India, this use case's main objective was to understand the action of air bag inflation under dynamic conditions.

"Microsoft Azure resources with UberCloud Containers and ANSYS LS-DYNA provide an excellent platform to develop and run accurate simulation models that involve complex impact physics."

High-performance cloud benchmarking

The HPC system used was a Microsoft Azure GS5 instance consisting of 32 cores, 448GB RAM, and running the Linux operating system. The airbag model was simulated using ANSYS LS-DYNA in an UberCloud container on the Microsoft Azure cloud platform.

The model evaluated the airbag's behavior, the rate at which the airbag opened and the stresses that developed on the airbag material. Different finite element models were developed for both fine and

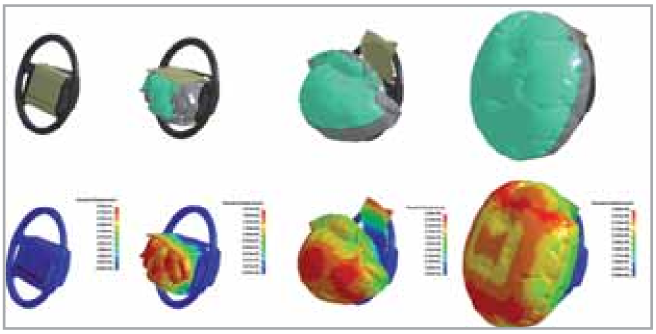


Fig. 1 - Plot of the Airbag's deformation. Top: Airbag opening sequence. Bottom: Contour plot of the steering and airbag assembly.

coarse mesh and the model data was submitted to the ANSYS LS-DYNA container. The time taken to solve the model with different mesh intensities was captured to benchmark the performance in solving high-density mesh models. The boundary conditions, solution algorithm, solver setup and convergence criteria remained the same

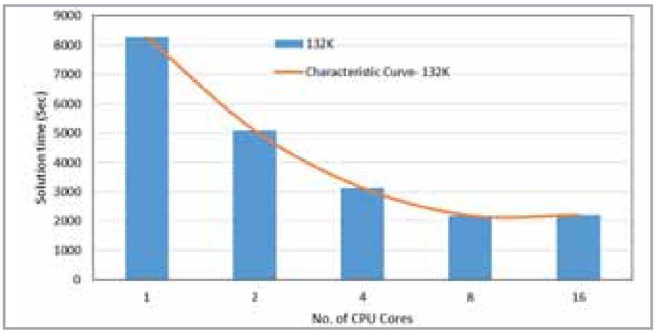


Fig. 2 - Solution time with a different number of cores for a model with 132K elements.

for all models. Fig. 2 shows the comparison of the solution times over a different number of CPU cores for a model with 132K elements.

Challenges

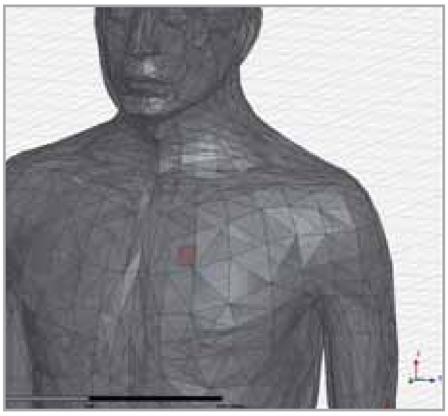
The technical complexity of the application and the very short time available to run all the simulations represented the greatest challenges in this project. This meant it was necessary to perform trials with different mesh-density models to accurately capture the airbag behavior. The finer the mesh, the better the accuracy but the greater the simulation runtime, and hence the challenge was to complete the simulation within the stipulated timeline.

Benefits and recommendations

1. The HPC cloud computing environment with ANSYS Workbench and LS-DYNA made model generation easier and drastically reduced the processing time, while result viewing and post-processing time were also dramatically decreased by the ANSYS / Azure / UberCloud HPC set-up.
2. The computation time required to run a single simulation with a fairly fine mesh (~132K elements) is substantial and is nearly impossible to achieve on a normal workstation. The HPC Cloud enables very fine mesh models to be solved and drastically reduces the simulation time to about 30 minutes.
3. The experiments run in the HPC Cloud clearly demonstrated that setting up and running the simulations remotely in the cloud was straightforward. The fact that the different simulation setup tools were already pre-installed in the HPC container meant that the user could directly access all the tools and the simulation software without need to run any installations first.
4. Access to the HPC Cloud was very simple and did not require any specific software to be installed. Since access take place by using VNC Controls in a Web browser, the whole user experience is similar to accessing a website through the browser.
5. UberCloud's ANSYS containers enabled smooth project execution with easy access to cloud server resources. The container dashboard helped monitor system performance and usage.
6. Microsoft Azure with UberCloud containers greatly simplifies advanced computational experiments that have high technical challenges and complex geometries and physics, which cannot be solved on a normal workstation.

Simulation in the cloud of an implantable planar antenna with ANSYS HFSS

This use case by Ozen Engineering in California concerns implantable antennas used in medical devices for communication purposes. Designing antennas for implanted devices is extremely challenging: the antennas must be small, low-profile, and multiband. In addition, they must be able to operate in complex environments. Factors such as small size, low energy requirements, and impedance adaptation play a significant role in the design process. Although several antennas have already been proposed for implantable medical devices, an accurate model of the full human body has rarely been included in simulations.



"ANSYS HFSS in UberCloud's application software container provided an extremely user-friendly on-demand computing environment very similar to my own desktop workstation."

An implantable Planar Inverted F Antenna (PIFA) has been proposed for communication between medical devices implanted in the human body and external medical equipment. The main aims of this project were to optimize the proposed antenna after implant in the skin tissue of the human body model and to characterize the effects of the electromagnetic radiation on human body tissues and the distribution of the specific absorption rate (SAR). The simulations were performed using ANSYS High-Frequency Structural Simulator

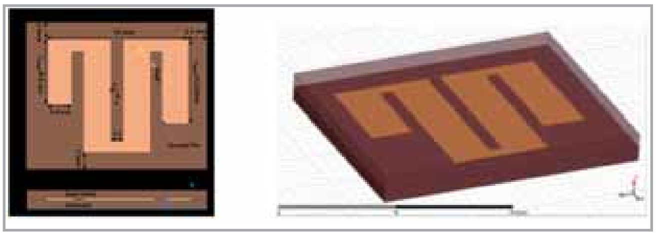


Fig. 4 - Top and side view of the PIFA (left); 3D view of the PIFA geometry in HFSS (right).

(HFSS), which is based on finite element method (FEM), together with ANSYS Optimetrics and high-performance cloud computing.

Antenna design and the ANSYS human body model

ANSYS provides adult-male and adult-female body models with variable geometric precision in the millimeter scale. Fig. 3 shows a general view of the models. The ANSYS human body model contains over 300 muscles, organs, tissues, and bones. For high frequencies, the body model can be electrically large, resulting in a huge number of mesh cells that make the simulation very time-consuming and computationally complex. The implantable antenna was placed within the skin tissue of the upper left chest where most implanted pacemakers and cardiac defibrillators are located, see Fig. 3. ANSYS Optimetrics and HPC features enable optimization iterations to be efficiently performed to accurately simulate the implantable antenna inside the human body model.

The antenna was simulated in ANSYS HFSS, an electromagnetic FEM solver. The top and side view of the proposed PIFA is shown in Fig. 4 (left), while the 3D view is shown in Fig. 4 (right). HFSS Optimetrics, an integrated HFSS tool for parametric sweeps and optimizations, was used to tune and enhance the antenna's performance within the ANSYS human body model.

Results and analysis

Fig. 5 shows the far-field radiation pattern of the proposed PIFA at 402 MHz. Since the antenna is electrically small and the human body provides a lossy environment, the antenna gain is very small

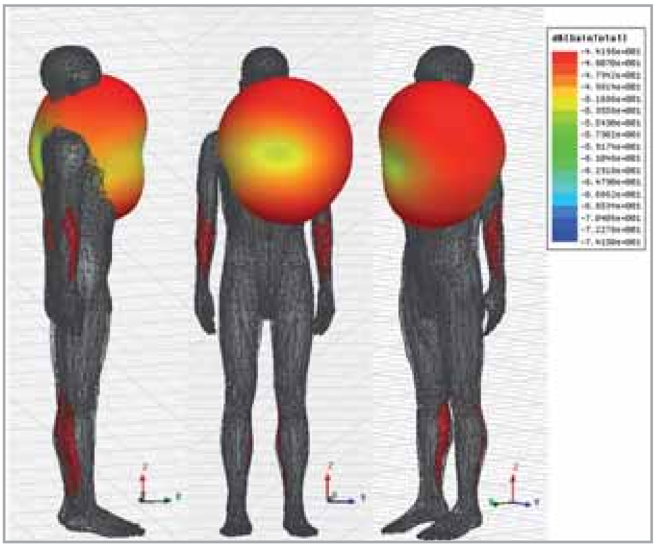


Fig. 5 - 3D Radiation pattern of a PIFA implanted within the human body model

(~-44 dBi) and the EM fields are stored reactively in nearby parts of the human body. A critical parameter was the electromagnetic power absorbed by the tissues surrounding the antenna within the human body model. Therefore, SAR analysis was necessary to evaluate the antenna's performance. SAR measures the density of the electromagnetic power absorbed by human body tissue. The SAR measurement makes it possible to assess whether a wireless medical device meets the safety limits. SAR is averaged over the whole body or over a small volume (typically 1-10g) of tissue. ANSYS HFSS makes standards-based SAR calculations.

Fig. 6 shows the 3D plot of the local SAR distribution for the male body model including the heart, lungs, liver, stomach, intestines, and brain. It can be observed that the SAR, which is significant in the upper left chest area, is relatively small. The PIFA's peak SAR is smaller than the regulatory limit for SAR.

Conclusions and recommendations

The modifications to the design and the tuning of the antenna's performance were studied with the implantable antenna placed

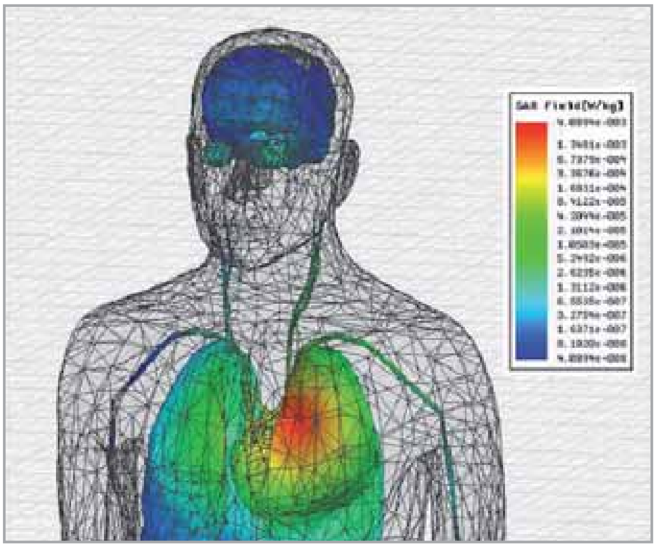


Fig. 6 - Localized SAR distribution on the upper side of male body model at 402 MHz.

within the skin tissue of ANSYS human body model. The resonance, radiation, and specific absorption rate (SAR) of the implantable PIFA were evaluated. The simulations were performed with ANSYS HFSS (high-frequency structural simulator) which is based on the finite element method (FEM). All simulations were performed on a 40-core Nephoscale cloud server with 256GB of RAM. These simulations were about four times faster than on the local 16-core desktop workstation.

ANSYS HFSS was packaged in an UberCloud HPC software container, which is a ready-to-execute package of software designed to provide the tools engineers need to complete their task. In this experiment, ANSYS HFSS was pre-installed, configured and tested, and operated on bare metal without any loss of performance. The software was ready to execute immediately without the need to install or configure the software or manage complex operating system commands.

"The students learned the ANSYS software extremely fast. It was really inspiring to see the many concepts they came up with."

"To see such a spirit among the students was the most inspiring moment in the entire project."

Ninth graders design flying boats with ANSYS Discovery Live in the Cloud

UberCloud Project no. 208 was carried out in collaboration with a class of 25 students from the ninth-grade of the Torsdad Middle School in Sandvika, Norway. They were assisted by their physics teacher, Ole Nordhaug, engineer Håkon Bull Hove from ANSYS channel partner, EDRMedeso, and HPC Cloud service provider, UberCloud. They used ANSYS Discovery Live on Microsoft Azure NV6 compute instances, each equipped with an NVIDIA Tesla M60 GPU.

The students used ANSYS Discovery Live for instant 3D simulations, tightly coupled with direct geometric modeling, to enable interactive design exploration and rapid product innovation. It is an interactive, Multiphysics simulation environment in which users can manipulate geometry, material types, or physics input and instantly see changes in performance. It allows users to test multiple design iterations in much less time. It is a great tool for CAD engineers who can control the physics of their different designs on demand.

Through a course called "Research in Practice", the students developed a combination of a boat and a sea plane. The students were free to innovate within certain parameters, such as a predefined height, width and length. To win the competition, the students had to develop the best overall design, which required attention to many different aspects of boat and aircraft design. At the end of the semester, all the boats were printed in 3D, and the students gave sales presentations to convince a panel that their concept was really the best.

3D modelling in ANSYS Discovery Live

The students designed a 3D model of the boat in ANSYS Discovery Live. None of the students had any previous experience with CAD modelling or simulation, but they faced the challenge squarely. Perhaps the most difficult task in the project was to prove that the boats could actually fly. To do so, the students performed aerodynamic analyses of the wing, examining both lift and drag. Their simulations not only showed that the wings had enough lift, but also provided an excellent demonstration of the physics.

Using UberCloud on Microsoft Azure

UberCloud supported the course students throughout the entire three-month design and simulation project. Each Wednesday, the company provided ten ANSYS Discovery Live environments on ten Azure NV6 Windows compute nodes, each equipped with 6 Intel Xeon E5 compute cores, 56GB, and an NVIDIA Tesla M60 GPU to accelerate real-time remote computing and visualization. The cloud resources were located in Microsoft's Azure datacenter in Amsterdam, which the students were able to access instantly via their web browsers with their login and password.

Conclusion

This middle-school project with 14-year old students and their physics teacher in Norway is just another impressive demonstration of the current trend towards more user-friendly application software, combined with the extremely fast and easy-to-use (GPU-equipped) HPC Cloud infrastructure from Microsoft Azure, which puts it at everyone's fingertips. It represents a big step towards "democratizing" high-performance computing and engineering simulation.

References

[1] Wolfgang Gentzsch, Engineering simulations using ANSYS in the cloud. EnginSoft Newsletter, Summer 2019. https://www.enginsoft.com/assets/pdf/newsletter/newsletter2019_02.pdf

[2] DOWNLOAD ANSYS UberCloud Case Studies, https://www.theubercloud.com/ubercloud-compendiums

For more information
Wolfgang Gentzsch - UberCloud
wolfgang.gentzsch@TheUberCloud.com

Fig. 7 - The winners from Torsdad Middle School in Sandvika, Norway, with their flying boat.

Optimization (Synthesis) of Tolerances to minimize production costs for a section of fuselage

EnginSoft and Leonardo collaborate on the Innovative Aeronautic Primary Structures (SPIA) project

SPIA

Enrico Boesso[1] and Salvatore Cascella[2]
1. EnginSoft - 2. Aircraft Division of Leonardo

The SPIA Project

The activity described in the article was developed within the framework of the SPIA (Innovative Aeronautic Primary Structures) funded research project. This project aimed to investigate innovative structural solutions (both from the design and manufacturing perspectives) to be implemented on the empennages and rear fuselage of a regional aircraft. This was achieved by starting from the material properties and using a scale-up approach, through the design, manufacture and testing of details and sub-components, up to the realization of fully-representative prototypes. Another objective was to improve the performance of the manufacturing process through the simultaneous design, development and testing of smart factory method technologies. The aim of the "Manufacturing Optimization" was to realize an integrated multidisciplinary solution based on technologies such as digital mock-up (DMU), virtual reality and digital manufacturing to support simulations and production processes, to be integrated with the company systems currently in use.

Product Requirements (Key Performance Indicators – KPIs)

The prototype is a small-scale section of the fuselage representing an assembly of metal and composite parts (see Fig.1). A set of key functional characteristics (KC) was identified to achieve the performance, assembly and installation objectives for this prototype. Each KC has its own dimensions (KPIs) which must first be validated and then verified against target values (T) and specification ranges (TOL) as defined at product level to ensure functionality. A product whose KC meets these requirements is a product that will meet the performance expectations of all stakeholders in the Product Life Cycle.

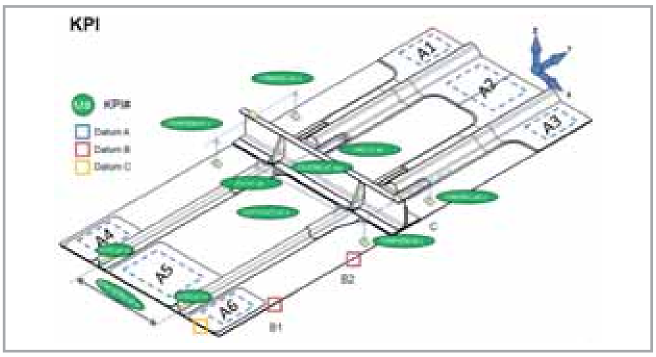


Fig. 1 - Prototype and geometric definition of the functional requirements subject to the calculation of tolerance propagation.

Tolerance analysis

Tolerance analysis is the initial phase of optimization. Specifically, it is the moment when the manufacturing process is validated in terms of the expected functional specification, which is derived from the desired product performance.

In the analysis, the design hypothesis is represented by 3D CAD models (Model Based Definition). From this first hypothesis it is possible to build a virtual model to evaluate the propagation effects of the variations in the assembled parts and their impact on the forecast specification ranges. Geometric models and drawings (dimensions and tolerances) are the input data for the propagation calculation.

Using appropriate indices (process capability Cp, Cpk, DPMU - Defects Per Million Units) it is possible to verify the KPI output's degree of compliance with the specification values (T, TOL). After the validation analysis and by reading the process indicators (in particular the DPMU defect), it is possible to establish whether or not an assembly process meets the product performance requirements. If not, the contribution coefficients and sensitivity are analyzed to determine the type of corrective action that should be taken, to decide both whether and where to intervene and modify the dimensions/tolerances to achieve the desired quality objectives.

The sensitivity analysis is used to identify the nominal dimensions with the greatest impact on the average value of the statistical distribution representing the individual functional requirement which is expected at the time of assembly.

The statistical contribution analysis is used to identify the tolerances with the greatest impact on the statistical dispersion around the average value of the individual functional requirement.

For complex products, however, corrective actions are not easy to identify due to the fact that the number of variables (nominal dimensions and tolerances) is very high and that these variables are often strongly correlated, in the sense that a dimension or a tolerance can simultaneously impact on multiple requirements in a discordant manner.

With regard to the calibration of the average: since the height of a part can have a different effect on different assembly requirements, it is not always possible to perfectly calibrate the assembly output to the design ratings. If we also add to this the considerations related to the cost of obtaining a certain precision (in general, the rule is that the lower the tolerance, the higher the cost of satisfying it will be, with exponential growth towards the

nominal condition of zero tolerance), it is easy to understand that finding the "best" compromise between quality and costs is, euphemistically speaking, very difficult with a manual "trial & error" approach.
With regard to reducing variability, in fact, it is well known that improving the precision of production requires major investments in the entire manufacturing chain. Furthermore, some variabilities cannot be improved because of the inherent limitations of the technology.
Tolerance analysis is a challenge that has neither an ideal nor a single solution. Rather, there are a number of trade-off possibilities, each with its own pros and cons. The choice of the best solution is subjective and is the result of the company's objectives based on the constrained resources that can be invested in the development of the project.
The analysis of the original project, submitted as part of the SPIA research, showed unsatisfactory results for almost all the KPIs.
Even after the calibration and correction of some of the nominal dimensions, which made it possible to optimize their disposition on the drawing, the variability of the manufacturing processes (in particular of the composite parts) still required some corrective actions to be taken in the manufacturing processes to improve their precision.
The decision as to which processes to invest in to implement the selected cost/opportunity trade-offs necessitated a dedicated optimization phase.

Optimization (synthesis) of tolerances

The optimization was performed by integrating the Cetol 6σ tolerance analysis software into a recursive loop of simulations managed by modeFRONTIER. The workflow created for the execution of the automatic optimization foresees:

- defining the input variables and their parameterization;
- defining the flow of operations for the automatic execution of Cetol 6σ;
- inserting the calculation and the evaluation of the cost function into the workflow,
- defining the output variables, the qualitative constraints applied to them and the objectives of optimization.

The synthesis of the tolerances consists of "ascending" the error propagation model constructed during the analysis phase. First of all, the quality level to be obtained in the final product is set and then, by

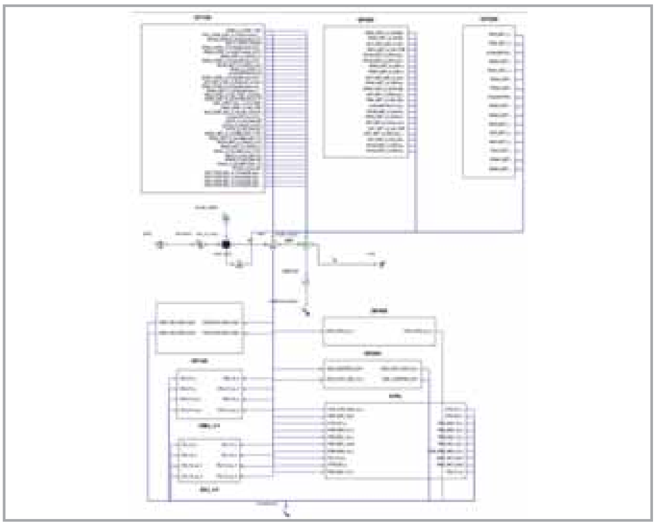


Fig. 2. Optimization workflow in modeFRONTIER.

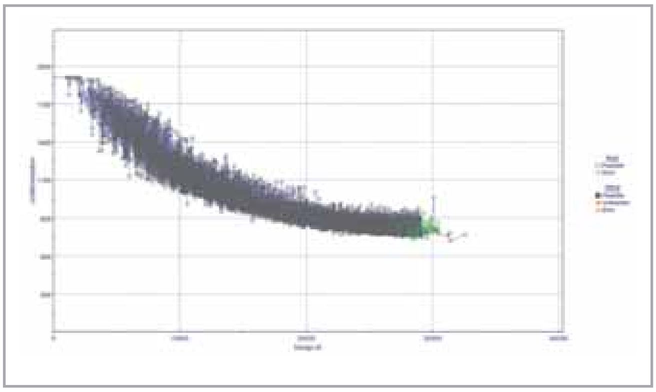


Fig. 3. Total cost as a function of the number of "feasible" designs. Progression of the optimization and minimization of the total product cost.

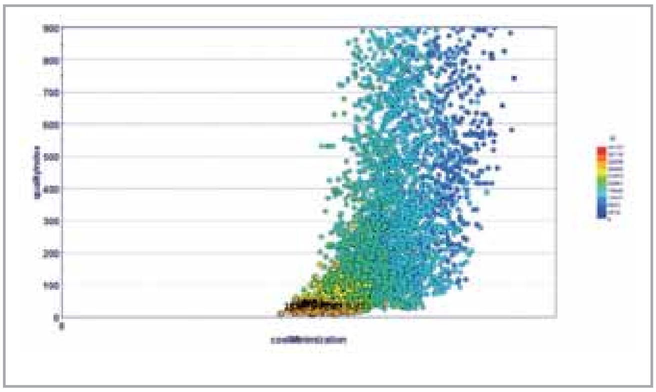


Fig. 4. Bubble chart of the two optimization objectives for "feasible" designs only. The colored scale represents the progress of optimization.

means of an automated recursive procedure, the individual tolerances are allocated to obtain satisfactory values for the quality indices. During this automatic iterative process, the production cost (suitably estimated by means of cost functions) is also calculated and minimized to obtain the desired final quality at the lowest possible production cost.

Conclusion

A practical application of the methodology has shown that it is able to achieve the objectives for which it was developed: to define sufficient precision only where it is actually necessary to obtain a final product of predetermined quality (number of percentage deviations).

This objective was achieved by implementing a numerical, automatic and recursive model using the Cetol 6σ CAT (Computer Aided Tolerancing) software which allows the calculation of the impact of the propagation of the design tolerances on the final quality of the product, integrated into a workflow created with modeFRONTIER. This integration allowed the practical application of the "synthesis of tolerances" (as opposed to the simple "analysis of tolerances") which involves establishing the final desired product quality and then allocating the individual tolerances in the project that allow it to be obtained.

For more information
Enrico Boesso - EnginSoft
e.boesso@enginsoft.com
or visit: www.dtascarl.org/progettidta/spia/

Optimizing turbochargers for high-performance engines

German-funded industry collaboration between business and academia creates comprehensive workflow

by Friedrich Fröhlig and Nicolas Lachenmaier[1], Stefan Harries and Carsten Fütterer[2], Johannes Ratz[3], Kathrin Wendl and Thomas Hildebrandt[4]

1. MTU Friedrichshafen GmbH - 2. Friendship Systems AG - 3. Technical University of Darmstadt - 4. NUMECA Ingenieurbüro

This article presents a project that created a fully-automated multi-objective and multi-disciplinary optimization approach that improves the efficiency of turbochargers for high-performance engines with regard to map width, mass, inertia, center of gravity, service life and the eigenfrequencies. The approach joins major software systems – CAESES®, FINE™/Turbo, STAR-CCM+ and modeFRONTIER – in streamlined workflows at MTU Friedrichshafen GmbH. Compressors and turbines were parametrically modeled, then they were numerically tested for aerodynamic and structural behavior, formally optimized and comprehensively assessed. The new approach identified considerable improvements that could be realized even for very good baseline designs. These improvements result in performance and cost benefits.

The need to generate power and propulsion sustainably has dramatically increased over the last few years. While it still remains unclear how society will transform to sustainably using available resources, it is mandatory to continuously improve the currently available solutions to the highest possible level. To this end, parametric optimization is growing steadily in the turbomachinery sector.

MTU Friedrichshafen already undertook the first fully-automated aerodynamic optimization campaign for a compressor wheel in 2012, and considerable achievements in efficiency and pressure ratios were realized. However, when it came to structural mechanics and service life calculations, the wheels that had been optimized for aerodynamic performance often proved to be infeasible. Furthermore, in those early stages, several other important aspects like inertia and eigenfrequencies were not yet correctly included. Unfortunately, subsequent attempts to manually tune the aerodynamically optimized wheels to acceptable values for stress, service life, inertia and eigenfrequencies repeatedly nullified a significant portion of the aerodynamic improvements. MTU therefore concluded that a purely aerodynamic optimization was unfortunately not a real optimization. Nonetheless, in view of the ever-growing market demand and regulatory requirements for high-performance engines such as high overall engine efficiency (CO2) and low NOx, SOx CO and hydrocarbon emissions, all engines and all their components must be thoroughly optimized (see example in Fig. 1).

A decisive engine component is the turbocharger, which must be well designed and suitably selected. This stimulated the desire to establish and apply fully automated, multi-objective and multi-disciplinary optimization for turbomachinery. When considering the complexity

Fig. 1 - MTU 16V BR4000 L64 Gas Genset

of developing a robust and reusable workflow that incorporated all key aspects, MTU decided that it would be advantageous to involve a consortium of experts. As a result, know-how in robust variable geometry, automated optimization, robust and high-quality meshing, high-fidelity computational fluid dynamics (CFD), finite element analysis and computational structural mechanics was brought together in the German state-funded GAMMA R&D project with four partners from industry and academia:

- MTU Friedrichshafen GmbH (turbine workflow [1], compressor workflow, optimization, testing and overall project management),
- Technical University of Darmstadt's Institute of Gasturbines and Aerospace Propulsion (GLR) (optimization workflow compressor [2]),
- NUMECA Ingenieurbüro (CFD, automated meshing, optimization), and
- FRIENDSHIP SYSTEMS AG (geometric modeling and parameterization, design space reduction [3]).

Turbochargers for power generation and propulsion

Natural gas engines are considered an important intermediate step for sustainable power generation. The amount of emitted carbon dioxide is directly proportional to the fuel consumption which in turn affects the lifecycle costs. One of the biggest levers to raise the overall engine performance is to improve the efficiency of the turbocharger. By a rule-of-thumb, a 1% rise in turbocharger efficiency increases engine efficiency by about 0.1%. Apart from this positive impact on the environment, an increase of 0.1% in the overall efficiency of the engine, for example for an MTU series 4000 16V gas engine (see Fig. 1), will reduce operating costs by several tens of thousands of euros per year.

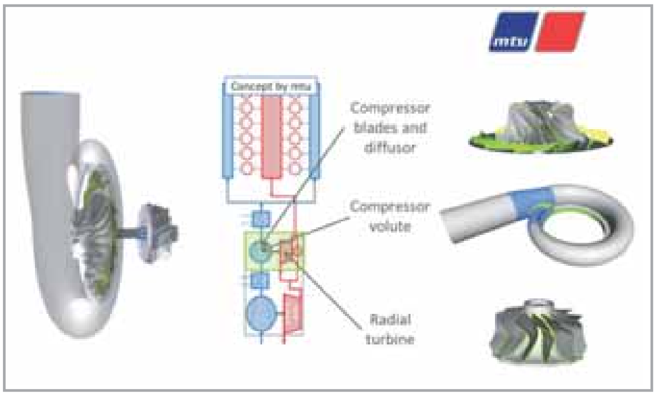


Fig. 2. Components of a turbocharger for a high-performance engine

While efficiency is the primary objective in stationary power generation, the situation in the propulsion sector is more complex. There are many more boundary conditions that must be considered. Efficiency is also very important, but map width, mass, inertia, center of gravity, service life and the eigenfrequencies of the compressor and turbine blades also need to be optimized, taking into account the limitations of available space and burst energy. A design change to improve one objective typically worsens another (Pareto optimality). The overall goal in turbocharger optimization is, of course, to find the best compromise for the specific application. For this reason, dedicated workflows have been developed for turbines and compressors. These are used to optimize compressors and turbines (see Fig. 2) both for two-stage gas and marine engines.

Compressor

The centrifugal compressor, driven by the turbine, increases the static air pressure in front of the engine. Therefore, more oxygen is provided for combustion in the cylinder, resulting in increased engine power. Typically, a centrifugal compressor consists of an impeller, a diffuser with or without vanes and a volute (Fig. 2). The impeller increases the kinetic energy of the fluid, while the diffuser and volute convert that kinetic energy into potential energy, i.e., static pressure. From a design point of view, there are two main objectives: firstly, the efficiency of the centrifugal compressor must be as high as possible, since the efficiency of the compressor has a direct influence on the overall efficiency of the engine. Secondly, the performance map should be as broad as possible since good transient response yields faster engine acceleration. In addition, the structural integrity of the impeller must be guaranteed. Impellers that are only optimized for aerodynamics often exceed the maximum permissible strains or have eigenfrequencies that are too low.

Workflow for compressor optimization

At MTU, the entire optimization process is now embedded and controlled by a process integration and design optimization (PIDO) system such as modeFRONTIER from ESTECO or FINE™/Design3D by NUMECA. Fig. 3 shows the workflow, highlighting the complexity of the set-up and the software systems involved: the centrifugal compressor geometry was parametrized in CAESES® by FRIENDSHIP SYSTEMS. Impeller and diffuser (see geometries provided in Fig. 2) were defined by hub and shroud contours in the meridional plane along with their blade angle distributions. Based on this, a 3D camber surface is created, and a thickness distribution is added to obtain the final blade geometry. Important design features such as a variable fillet radius, a nonlinear leading edge or a change in blade ruling could also be easily implemented in the geometric model. The volute was mainly defined by its inlet radius and the A/R ratio. While the parametric definition of the geometry was undertaken in the GUI mode of CAESES, instances of the geometry were created in the batch mode of CAESES during the optimization campaigns. After the geometry has been created, the mechanical constraints are checked. To this end, the solid material of the impeller was meshed with either HEXPRESS™/Hybrid from NUMECA or SimLab. Subsequently, the von Mises stresses and eigenfrequencies were calculated with CalculiX. The aerodynamic evaluation of the centrifugal compressor was performed with NUMECA's comprehensive software suite. For the fluid domain of all three components (impeller, diffusor and volute), a structured mesh was generated with NUMECA AutoGrid5™ and NUMECA IGG™, respectively. The Reynolds-Averaged Navier-Stokes (RANS) equations were solved with NUMECA FINE™/Turbo at three operating points to obtain information about the efficiency and width of the performance map.

Both the mechanical and aerodynamic results were then transferred to modeFRONTIER or FINE™/Design3D to start the next variant for generation, simulation and evaluation, Fig. 3.

Selected results

A total of 1 313 design variants were successfully evaluated for compressor optimization (see Fig. 4). The initialization was done by a design of experiments (DoE), which was created with a Uniform Latin Hypercube algorithm. The optimization was performed with the Fast MOGA II and PilOpt optimizer from modeFRONTIER. Alternatively, an ANN combined with a genetic algorithm in FINE™/Design3D could be used. The selected design shows an efficiency improvement of 2.2 percentage points and a performance map that was 5.3 percentage points wider than the baseline's. In addition, all mechanical constraints were met, which was not the case with the manually created baseline.

Turbine

The radial turbine converts the enthalpy of the exhaust gas into torque to drive the compressor on the same shaft. From a design point of

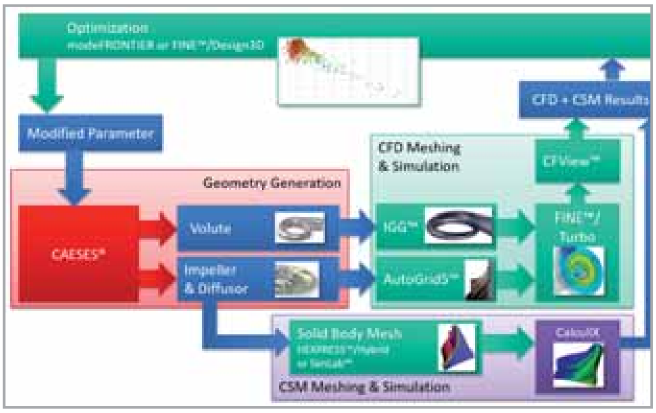


Fig. 3 - Optimization workflow realized within GAMMA and run at MTU for compressor design

view, the turbine development process is determined by several requirements: the most important is the structural integrity of the wheel during operation, which is affected by both high rotational speeds and intense temperature stress. Consequently, heavy yet robust metals must be used to cast the turbine. This leads to high mass and inertia of the rotor, deteriorating the transient response of the engine. Therefore, when attempting to maximize rotor efficiency to reduce the engine's fuel consumption, one must take into account minimizing the weight of the rotor and ensuring a certain life expectancy.

Similar to the compressor workflow described above, a setup for radial turbines was created using a slightly different chain of tools. However, the general framework is the same.

Workflow for turbine optimization

Once again, a parametric model of a radial turbine was created with CAESES®. The model contains more than 70 parameters. During the optimization campaigns, however, only 35 to 40 parameters were selected as free variables. The turbine body was constructed in three steps: starting from the meridian view, a camber line was described and finally complemented by a thickness distribution. Apart from these basic functions, the model offers several features that add value to the design process: the first are the automatically generated fillets on the hub line of the blade, while the second are the fully parametrized scallops between two blades to reduce the turbine's inertia.

Fig. 5 shows three representative variants for illustration. They were created completely automatically in CAESES® by changing the free variables of the parametric model. Based on the current geometry, the fluid is automatically meshed with Siemens' Star-CCM+ while the solid body is discretized in Altair's SimLab. A CFD simulation was conducted to calculate the efficiency of the rotor, and was supplemented by a conjugate heat transfer (CHT) simulation to determine a simplified temperature field for the solid body. Both this temperature field and the rotational speed of the turbine are used as boundary conditions for a finite element simulation of the solid model to determine the stresses and eigenfrequencies of the blade. In a final step, the mass and inertia of the model are calculated using a surface mesh of the solid body. The entire process is contained in and driven by modeFRONTIER to perform the optimizations. A schematic view of this process is shown in Fig. 6.

Selected results

In this case study, a design of experiments with 395 designs was carried out, followed by an optimization using the Fast MOGA II algorithm. A further 1 305 designs were evaluated in this way. The design database is shown in Fig. 7. The final design showed an improvement in efficiency of 1.5%, while inertia was reduced by 12% compared to the reference design. All structural constraints were fully met.

Conclusions

By combining dedicated systems for Computer Aided Engineering, advanced optimizations of turbochargers that lead to performance gains for high-performance engines both for power generation and propulsion can be realized. In a comprehensive R&D project, four partners – MTU, TU Darmstadt, NUMECA Ingineurbüro and FRIENDSHIP SYSTEMS – joined forces to establish sophisticated workflows that will facilitate the design and further optimization of compressors and turbines for MTU Friedrichshafen GmbH.

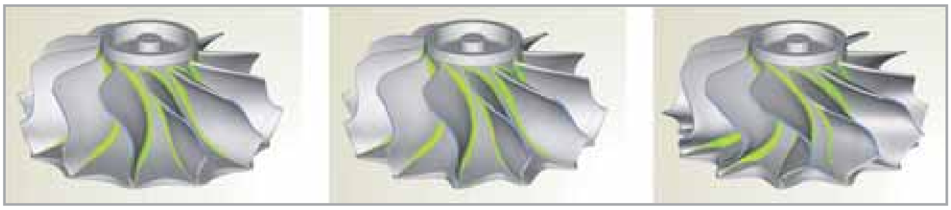


Fig. 5 - Variants of turbine geometry featuring the scallops created in CAESES®

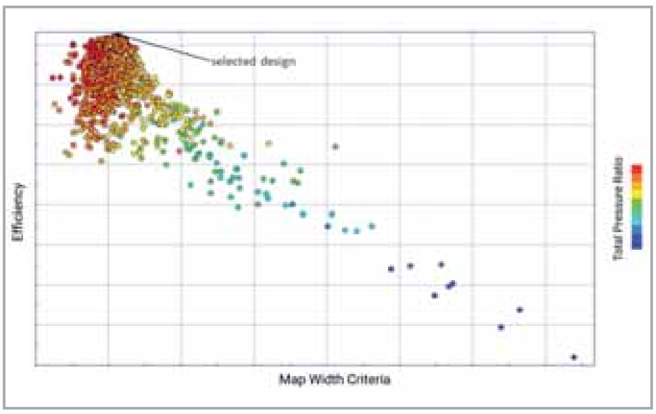


Fig. 4 - Selected results for compressor optimization

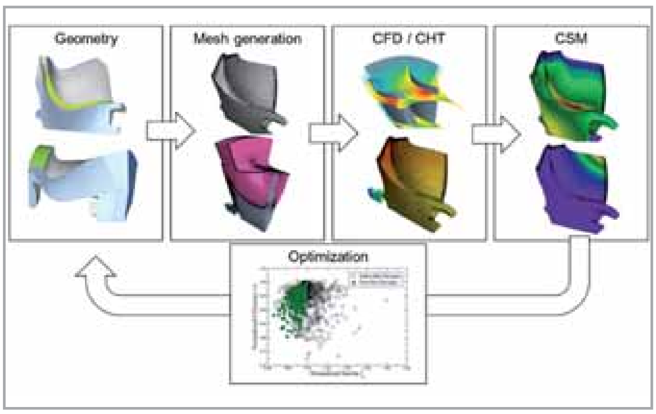


Fig. 6 - Optimization workflow realized within GAMMA and run at MTU for turbine design

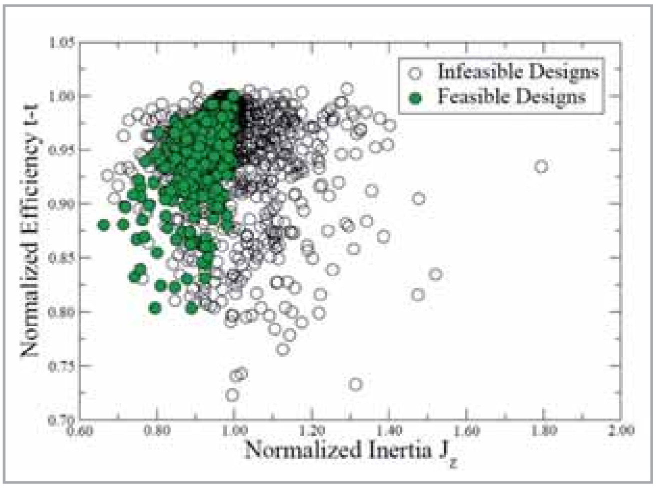


Fig. 7 - Optimization results for the turbine

Even though very good designs are already available at MTU, the new multi-objective, multi-disciplinary approach delivers turbochargers with higher efficiencies and wider performance maps, while reducing mass and moments of inertia and improving various other key aspects. Most importantly, no manual changes are required after the automated optimization campaigns, since all constraints are immediately taken into account. This not only saves time but also avoids the partial loss of some of the performance improvements identified during optimization at subsequent design stages.

References

[1] N. Lachenmaier; D. Baumgärtner; H.-P. Schiffer; J. Kech: "Gradient-free and gradient-based optimization of a radial turbine," Proceedings of 13th European Conference on Turbomachinery Fluid dynamics & Thermodynamics, April 8-12, 2019; Lausanne, Switzerland

[2] J. Ratz; S. Leichtfuss; M. Beck; H.-P. Schiffer; F. Fröhlig: "Surge margin optimization of centrifugal compressors using a new objective function based on local flow parameters," Proceedings of 13th European Conference on Turbomachinery Fluid dynamics & Thermodynamics, April 8-12, 2019; Lausanne, Switzerland

[3] E. Bergmann; C. Fütterer; S. Harries; J. Palluch: "Massive Parameter Reduction for faster Fluid-dynamic Optimization of Shapes," Int. CAE Conference and Exhibition, Vicenza, Italy, October 2018; see http://proceedings2018.caeconference.com/sessions.html#trans

Acknowledgements

Parts of the work presented here originate from the German R&D project GAMMA (Effiziente GAsmotoren für MaritiMe Anwendungen der nächsten Generation – Efficient GAs engines for MaritiMe Applications of the next generation, FKZ 03ET1469D). The project is funded by the Federal Republic of Germany, Federal Ministry of Economics and Technology on the orders of the German Bundestag.

For more information:
Stefan Harries - Friendship Systems AG
harries@friendship-systems.com

Status Report on the Competence Centers - Where are we?

SMACT Competence Center

MADE

After the approval of the final rankings by the Italian Ministry of Economic Development, between December 2018 and February 2019 the eight Competence Centers that are part of the Italian national plan for Industry 4.0, which later become Enterprise 4.0, were legally established.

The Competence Centers are legally constituted Public-Private Partnerships (PPPs) between universities, research centers, technology providers and enterprises. Their task is to enable small and medium-sized enterprises to acquire the skills to participate in the fourth industrial revolution currently underway. With total public funding of €73-million over three years, the Competence Centers will develop various specializations (both technological and sectorial) all aimed at supporting the digitalization processes of the Italian companies that want to make use of their services.

Since their establishment, the Competence Centers are now almost all at an advanced stage in their development: they have created an organizational structure and are about to launch the Calls for Proposals for IRISS (Istituto di Ricerca su Innovazione e Servizi per lo Sviluppo) projects with the public co-financing provided for by the Ministry of Economic Development. These are projects that will focus on innovation and services for economic development.

EnginSoft is deeply involved in the design of two of the Competence Centers:

1. MADE: Led by the Polytecnico of Milan, it involves four universities and 39 private companies as well as INAIL, the Italian National Institute for Insurance against Accidents at Work. This is the center of competence for the Manufacturing sector and will provide companies with services to digitalize their production processes to become Enterprise 4.0 companies.
2. SMACT: Led by the University of Padua, it includes the eight universities of the Triveneto region, two research organizations, the Chamber of Commerce of Padua and 29 private companies. This Competence Center's name is the acronym for the five technologies with which it deals, namely Social, Mobile, Analytics, Cloud and The Internet of Things, which will be applied in the sectors of agriculture and food, clothing, decor (furniture) and automation.

On 29 October 2019, SMACT will present its approach to assisting companies to tackle digitalization with maximum technical and operational effectiveness and will present three technology 4.0 live demos from the sectors it covers at the 2019 International CAE Conference. A Round Table discussion with the participating entrepreneurs and managers will follow. Visit www.caeconference.com to register and attend the conference and this session free of charge.

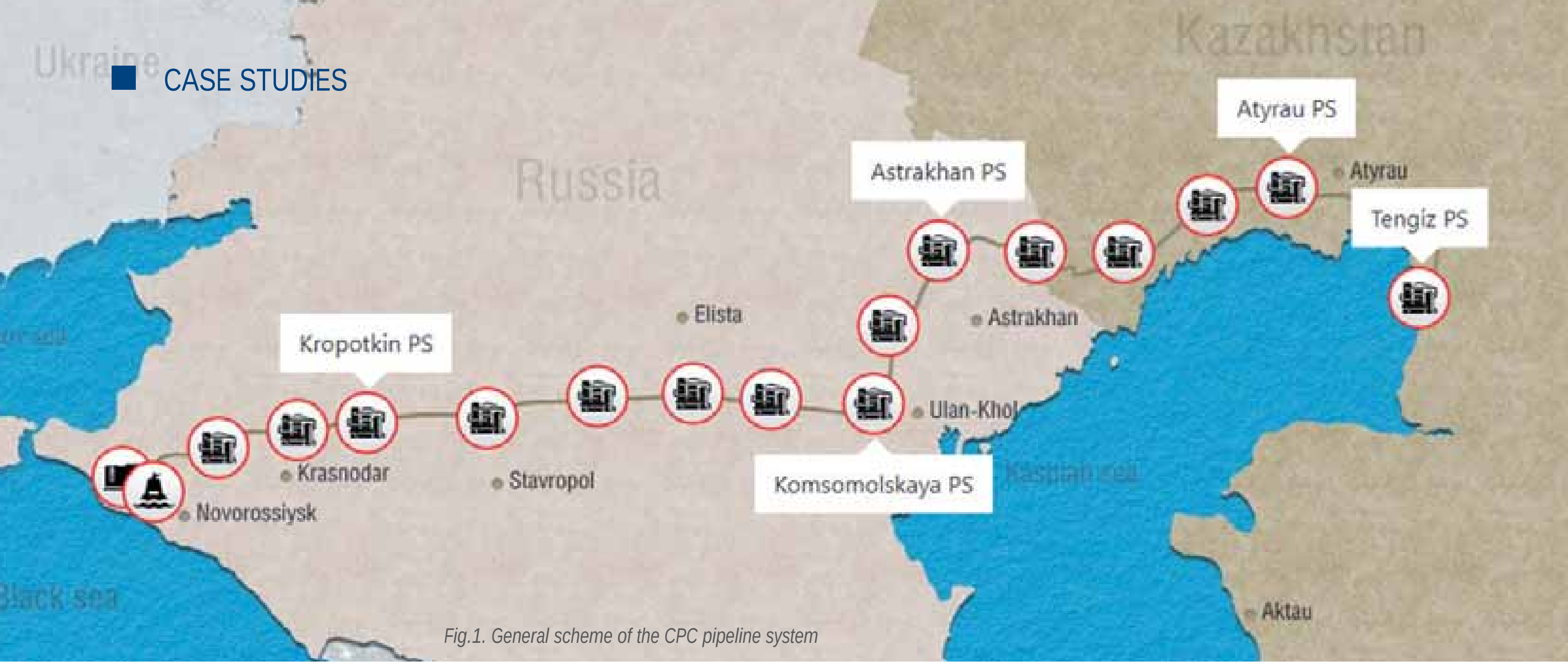


Fig.1. General scheme of the CPC pipeline system

PASS Software helps Caspian Pipeline Consortium to double pipeline capacity

Accurately modelled complex process piping network and all hydraulic elements to facilitate design and efficiency

From 2011-2017, the Caspian Pipeline Consortium (CPC) successfully implemented an ambitious pipeline capacity expansion project that allowed them to almost double their trunk oil pipeline capacity – from 35 to more than 60 million tons of oil annually. PASS/HYDROSYSTEM and PASS/START-PROF software played an important role in this project and proved to be effective tools to achieve success.

CPC (www.cpc.ru/en) is a major international project for the transportation of crude oil that includes the participation of Russia, Kazakhstan and leading international oil and gas companies (Eni International (N.A.), Chevron Caspian Pipeline Consortium Company, and Mobil Caspian Pipeline Company).

The project was created to construct and operate a trunk pipeline more than 1,500 kilometers long. The CPC pipeline system mainly collects crude oil from the large oil fields of western Kazakhstani and Russian producers and transports it to the CPC Marine Terminal on the Black Sea.

The CPC Expansion Project aimed to increase the CPC pipeline system's capacity up to 67 MMTA. The project envisaged the construction of new oil pumping stations, and the modernization of existing pumping stations with new equipment and process piping, as well as the extension of the tank farms at the CPC Marine Terminal, where PASS software played a critical role.

The project included a large number of hydraulic calculations of the process piping network with a complicated topology, which


Fig. 1 - Tank farm at the CPC Marine Terminal

were performed using PASS/HYDROSYSTEM. Firstly, it involved analyzing the new complex process piping network of the CPC Marine Terminal tank farm which interconnects the ten oil tanks with the Marine Terminal facility for loading tankers. The very detailed and precise 3D calculation model contained more than 200 piping branches with all their hydraulic elements, including the pipes, fittings and valves.

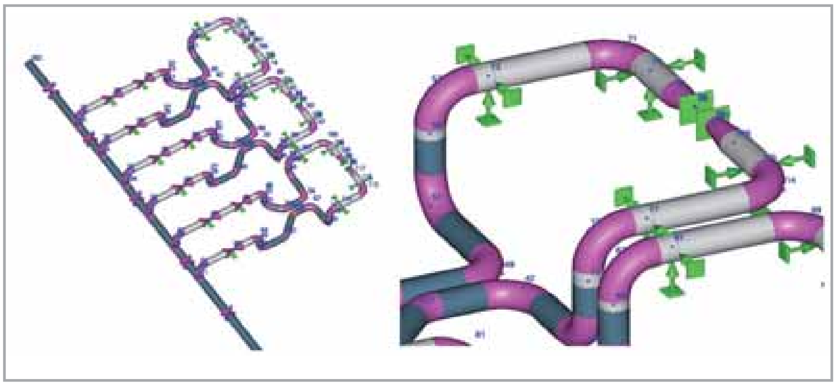


Fig. 5 - CPC Pump Stations' process piping calculation model and its fragment in PASS/START-PROF

The PASS/HYDROSYSTEM software made it possible to easily and quickly analyze the different operating modes of this complex system. The program uses color to identify operative and inoperative parts of the system, and shows flow-rate distribution, pressure losses and other calculation results. The software also helped to size all the control valves contained in the system to ensure optimal system operation in all operating modes.

In addition, the program made it possible to calculate the parameters of the fire extinguishing loop system within the marine terminal. This system consists of ten fire-fighting water supply loops; the PASS software was able to accurately select the fire pumps and fire hydrants based on the given parameters, which would have been impossible to achieve with manual hydraulic calculations.

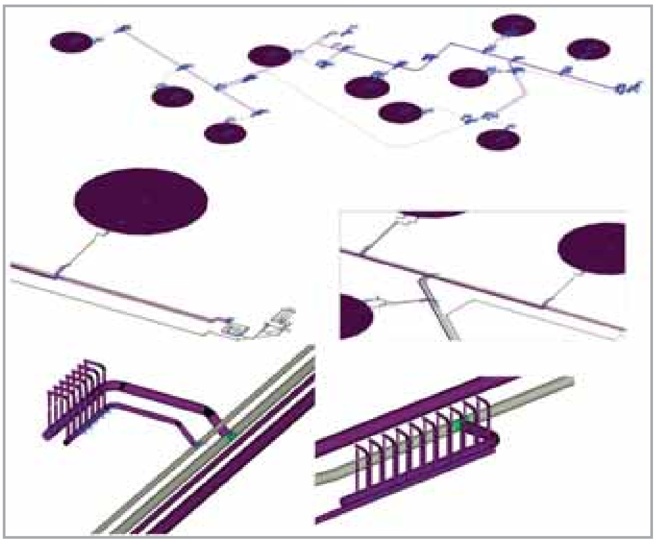


Fig. 3 - The CPC Marine Terminal tank farm piping network calculation model and some of its fragments in PASS/HYDROSYSTEM

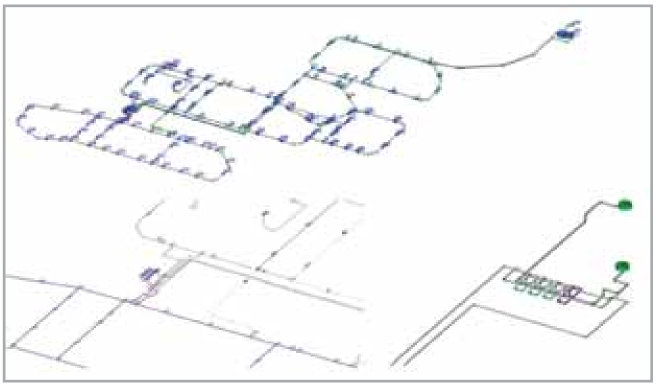


Fig. 4 - The CPC Marine Terminal's fire extinguishing loop system calculation model and its fragments in PASS/HYDROSYSTEM

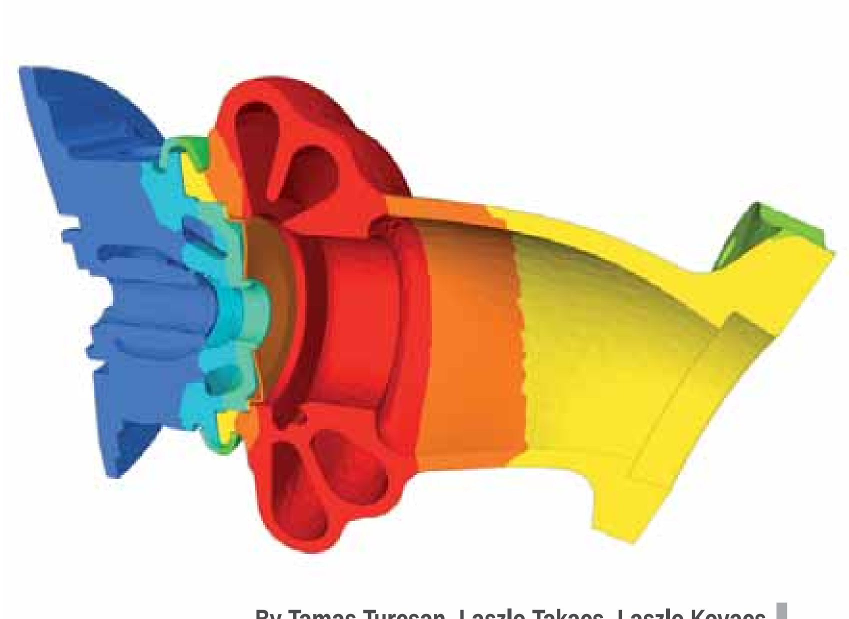
Both PASS/HYDROSYSTEM and PASS/START-PROF were used in the design of the new process piping of the CPC pumping stations. Specifically, piping stress analysis by PASS/START-PROF allowed the calculation of piping integrity and estimate loads on the piping supports, which allowed the designers to identify and select optimal support types and locations.

This project created reliable and efficient operating facilities at CPC. The PASS models developed within the project framework now enable the simulation of their various modes of operation, and allow them to consider some possible non-standard scenarios. The PASS Software significantly improved the quality of the engineering solutions and the resulting designs. It is also worth mentioning the high level of accuracy obtained with the PASS software analysis results when compared with test data obtained from the real assemblies and facilities.

Currently, the CPC has adopted and is implementing a debottlenecking program that is aimed to further increase pipeline capacity. This program is also actively using PASS software to identify efficient engineering solutions.

For more information:
Livio Furlan - EnginSoft
l.furlan@enginsoft.com
or visit: www.passuite.com

Innovation Driven Composite FEM Solutions at eCon Engineering



By Tamas Turcsan, Laszlo Takacs, Laszlo Kovacs
eCon Engineering Ltd

This article illustrates the way that innovation is driven by practical experience and market trends at eCon Engineering. The activities described in the article are primarily related to the calculation of fiber-reinforced polymer composites using the mechanical finite element method. Fiber-reinforced polymer composites have a broad spectrum of applications, but this article focuses on the characterization of the properties and failure criteria for separate layers based on simple material tests and the optimization of the composites' structure. By using these methods, more efficient composite structures can be designed and manufactured with better cost-performance trade-offs. The newly developed methods enable a greater understanding of the design principles and the possibilities of these materials and allow eCon to expand the range of its technical services.

Innovation and design

In the past few years, progress in the design of the fiber reinforced composites (FRP) has been driven by the capabilities of structural materials and, of course, by the utilization of composite structures. Efficiency can be enhanced by using the best possible design practices aided by finite element method (FEM) modelling, which is why fiber-reinforced composite FEM is increasingly affecting the industry. Attainable goals such as high stiffness, low structural weight, low risk of failure and other useful properties can be achieved by using appropriate FEM tools, but in most cases, these are not available on the market [1-3, 5].

Analytical method for ply specific properties

In addition to high strength-per-unit mass and corrosion and fatigue resistance, the heterogeneity of composites results in a high degree of uncertainty concerning the material's stiffness and strength. This uncertainty needs to be adequately addressed. The work presented here introduces a new concept that uses the raw surface deformation and load information from simple mechanical tests of multi-directional lay-up laminates and returns engineering constants of unidirectional (UD) layers. Since it is based on the backward use of the classical laminate theory (CLT), it has been called CLT-1. Based on the stiffness formulation used in CLT, all relevant in-plane engineering constants, such as moduli and in-plane Poisson's ratio can be evaluated in one step for each layer (see Fig. 1). This data can be directly inserted into the FEM preprocessing and can be useful in post processing for calculating failure criteria [1, 2].

The new method has been successfully proven by mechanical tests and simulations run in a FEM environment and has also been compared with the results of the conventional method. The approach has proven to provide faster progress and a more reliable shear modulus, as well as a more sophisticated probability distribution of all the constants, enabling the design of composite parts to a given level of deformation or failure probability [1, 2].

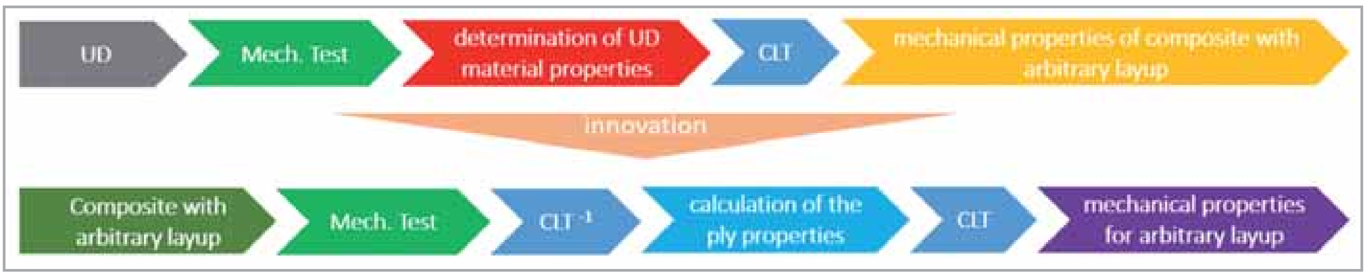


Fig. 1 - Estimation process of composite laminate stiffness – conventional (top) vs. CLT-1 method (bottom) [1]

About eCon Engineering Ltd

eCon Engineering Ltd, was founded in Hungary in 2002 with 70 engineers. With almost 20 years' experience in FEM, BEM and automated machine building competencies, the company solves various problems in the automotive, aerospace, energy, heavy duty, bus, composite and railway industries.

The 25 engineers in our CAE Services branch provide tailored solutions from meshing and model building, through collaboration with in-house CAE teams, up to complete simulation for development processes, across FEA, CFD, MBS and 1D simulations, with a wide range of CAE software, including ANSYS, Moldex3D and Cast-Designer.

In addition to FEM calculations, eCon provides special purpose machines and robot cells for the automotive and electronic industries and covers a wide spectrum of assembly technologies, with a special focus on increasing machine up-time and reliability.

Layer build-up optimization

The optimization is based on a design of experiments (DoE) (see Fig. 2a) which reveals the critical parts of the structure. According to the DoE, the structure is divided into sections that have a greater or lesser impact on stiffness at predetermined loads. The thickness and direction of the fiber reinforcement are the parameters for creating the variant. Parameters were created within feasible limits and, of course, reasonable steps were taken between each design point. As a result of the large variants, the trade-off between mass and stiffness was established and the best possible layer build-up was determined for each part of the component (see Fig. 2b).

After the optimization process, we were in a position to exploit the most advantageous properties of the composite materials and achieved a weight reduction of up to ~20% while incrementally increasing the stiffness by 10-15%. This also represents a significant saving in manufacturing and operating costs.

Conclusions

It can be stated that advanced composite FEM methods can save a great deal of development time and budget with better input parameter derivation and more efficient material use. On the one hand, an algorithm was introduced that derives the ply specific properties of the layers from simple mechanical testing of the multidirectional laminate. As result of these more accurate input parameters, at the end of the FEM calculations, more accurate results can be obtained in critical areas such as ply-wise stresses and strains, and composite laminate failure criteria. On the other hand, we introduced an advanced optimization method for better utilizing orthotropic composite materials. This process can significantly reduce both weight and manufacturing costs.

References

[1] Kovács, L. and Romhány, G.: (2018) "Derivation of Ply Specific Stiffness Parameters of Fiber Reinforced Polymer Laminates via Inverse Solution of Classical Laminate Theory", Periodica Polytechnica Mechanical Engineering, 62(2), pp. 158-164. doi: https://doi.org/10.3311/PPme.11846.

[2] Kovács L., Fodor B.: (2019) "Analytical Method to Obtain Ply Specific Engineering Constants from Simple Material Tests of NCF Composite Parts" NWC NAFEMS World Congress 2019 June 17-20 Quebec City, Canada

[3] Kovács L., Romhány G.:(2018) Effect of variability of mechanical parameters on the deformation behavior and failure of fiber reinforced plastic materials. in 'ECCM18 - 18th European Conference on Composite Materials Athens, Greece. 2018.06.24-2018.06.28.,

[4] Introduction into Design of Experiments DOE with HyperStudy, 1st Edition; Released 01/2017

[5] Takács L.: (2019) Cost-effective partitioning of a composite bus structure from the perspective of adhesive joints, 18 European Conference on Composite Materials -ECCM 18-25-28 June, 2018, Athens, Greece

Other activities

In addition to the methods presented, several best practices are used at eCon Engineering to calculate composite structures [3, 5]:

- Cost-effective separation process of bonded parts,
- Smart FEM mesh-generating processes,
- First ply failure model parameter fitting algorithm,
- FEM solutions for the stress-dependent properties of composites,
- Composite manufacturing defect estimation in FEM,
- Finite element solutions for composite fatigue.

For more information
Gabor Kiglics - eCon Engineering
kiglics@econenegineering.com

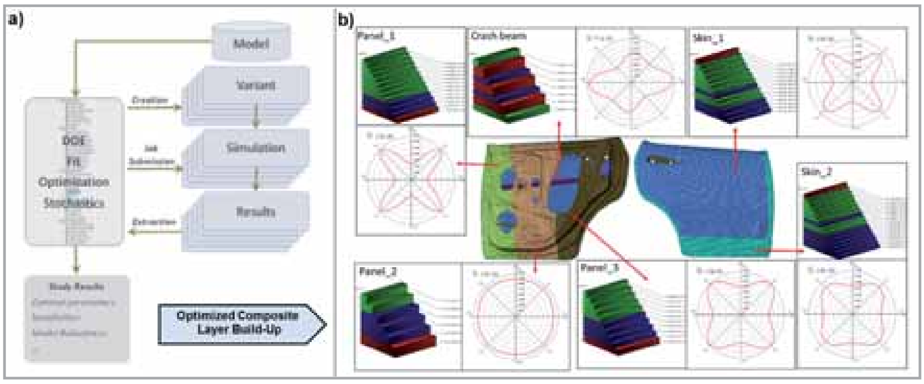


Fig. 2 - The optimization process (a) and the results of layer build-up process (i.e. the build-up of the layers and angle-dependent stiffness of the sections) presented in a side door of the car (b) [4]

Lattice optimization with GENESIS and GSAM

New capability accelerates analysis time and enables determination of optimal lattice dimensions

by Juan Pablo Leiva
Vanderplaats Research & Development Inc.

Additive manufacturing makes it possible to fabricate complex designs. Lattice optimization enables the design of structures that are structurally efficient. With GENESIS or GSAM optimization tools, you can maximize additive manufacturing's potential.

Additive Manufacturing (AM) is an emerging manufacturing technology in which a structure is typically fabricated by adding material in a layer-by-layer fashion, as opposed to the 'subtractive' traditional machining, in which material is removed from a raw block. With AM, it is possible to print lattice structures that would be difficult or impossible to fabricate with conventional manufacturing. The increased freedom in part shapes makes it possible to fully exploit optimization to design parts with minimal weight and maximal stiffness/performance.

Lattice structures have many benefits such as good strength-to-weight ratio, excellent shock absorption and impact protection behavior, a high surface area which can help to dissipate the heat, and more. However, the details in lattice shapes can create difficulties in CAD modelling and FE meshing, thereby making it more challenging to analyze and optimize the lattice structure. The latest release of the structural optimization software, GENESIS, allows solid elements to be used to simulate lattice structures. GENESIS uses homogenized materials that represent either a built-in lattice pattern or a user-supplied lattice pattern. Sizing and/or topometry can be applied to the homogenized lattice properties on solid elements to directly optimize the diameter/thickness of the lattice structures. The GENESIS lattice optimization workflow is shown in Fig.1.

Homogenization

For a given type of lattice cell and volume fraction, homogenization is used to obtain the equivalent material properties for solid elements. For triple symmetric cells, CAE material testing is performed for two load cases, axial and shear, as shown in Fig.2. For non-symmetric cells, additional load cases are used but are not shown here. The equivalent material properties are computed for a series of volume fractions, and a curve is fitted based on the test data (Fig.3). The fitted curve for material properties can be used in subsequent analysis and optimization. With homogenization, it is possible to analyze and optimize lattice structures without creating the actual lattice geometry and mesh for a macroscale model. The homogenized structure is much faster to analyze and optimize due to the reduction of the problem size.

In the current version of GENESIS, there are three built-in lattice types: cubic edges, edges with space diagonals and double pyramids. A user can also specify their own homogenized lattice properties with the user-supplied option (LATMAT). Note that the lattice type is not limited to a bar-type lattice. Other types of lattices can also be homogenized and optimized using GENESIS's lattice optimization capability. The software assumes lattice material properties to be orthotropic.

Lattice optimization using sizing/topometry

Through homogenization, we obtain the equivalent material properties of a given type of lattice and apply it to solid elements. In the latest release of GENESIS, a user can directly optimize lattice diameter/thickness using sizing or topometry optimization. Here the lattice cell size is pre-defined, and the designable quantity is the lattice diameter or thickness. The goals of lattice optimization and topology optimization are quite different. In topology optimization, we try to either keep material or remove material completely from the design region, so any areas with intermediate stiffness are unwanted. In lattice optimization instead, any diameter/thickness (within an allowable range) is acceptable, which actually enables the design to fully exploit the advantage of the lattice structure.

In the example below, a cylindrical column has a pressure load on the top (Fig.4(a)). The load path is typically not well defined for such structures, which makes this a good candidate for using lattice design. We start with a uniform lattice diameter, with the goal of maximizing the stiffness without increasing the mass. Topometry optimization redistributes the mass to the most critical location. Fig.4(b) shows the distribution of the optimized lattice diameter/thickness. The stiffness of the structure is improved by 21.9%.

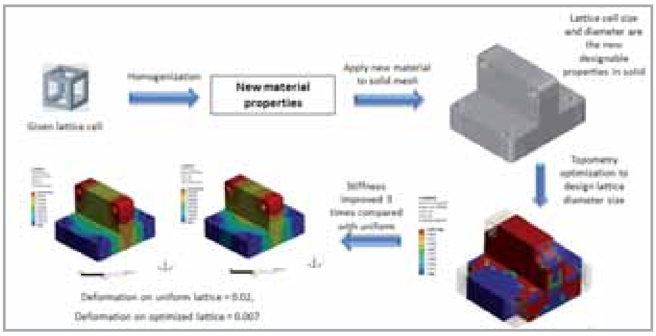


Fig. 1 - Lattice optimization workflow

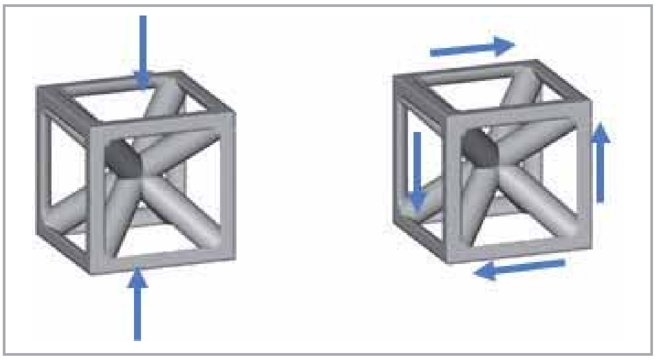


Fig. 2 - Uniaxial test and pure shear test

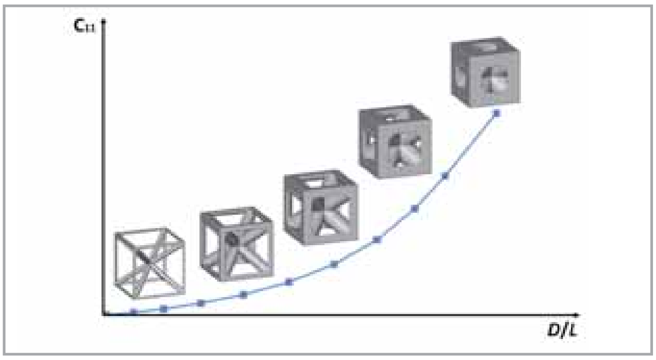


Fig. 3 - Material matrix coefficient vs diameter fraction

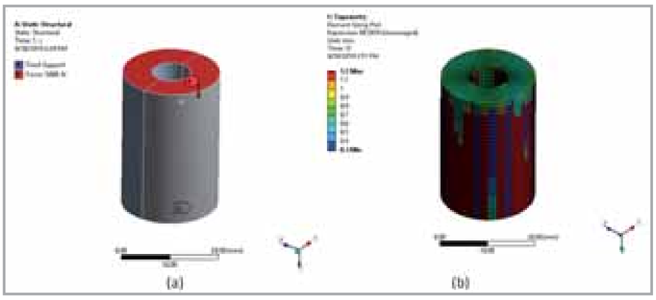


Fig. 4 - Topometry optimization of a solid part with homogenized lattice properties

The above example was created and solved using GSAM. GSAM is an ACT extension for ANSYS that uses GENESIS for optimization and ANSYS Mechanical for model creation and to display the topometry results. Design Studio was used for further processing.

Post-processing

After optimization, the result can be exported as an explicit lattice model using Design Studio for GENESIS (Fig.5). There are three export options:

1) Beam finite element mesh, which can be used for further analysis.
2) Graph (nodal locations) and Report (dimensions) files for Materialise 3-matic software, which can be used for printing.
3) STL model, which can be used for printing.

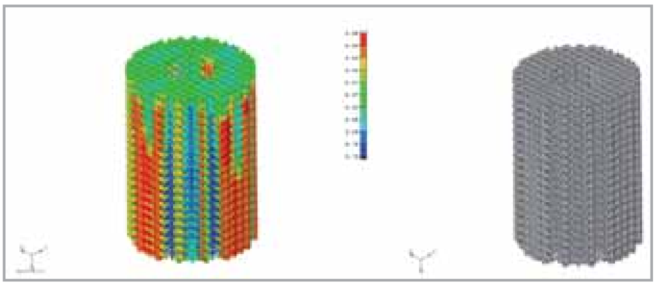


Fig. 5 - Lattice representation of optimization result

About Vanderplaats Research & Development, Inc.

Vanderplaats Research & Development, Inc. (VR&D) strives to provide the best technology, software, staff of experts and client support in the optimization world. Privately held with offices in North America and distributors around the world, VR&D was founded more than 30 years ago by one of the best-known experts in the optimization community, Dr. Garret Vanderplaats. VR&D has a track record for consistently delivering a competitive advantage to customers in a broad range of industries. More information on VR&D can be found at http://www.vrand.com/.

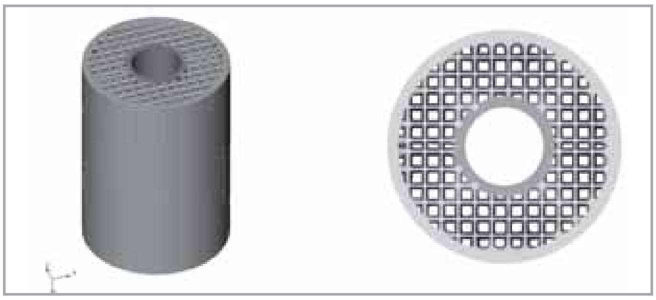


Fig. 6 - Lattice model with external skin kept

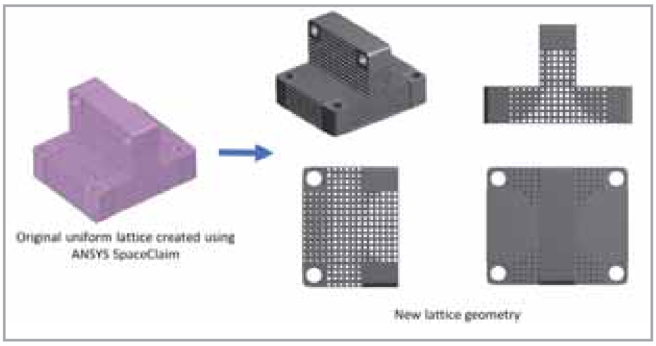


Fig. 7 - Map optimized diameter size to new lattice geometry

In creating a new lattice model, it is advisable to maintain the external skin of the original structure. This generates a more conservative structural design (Fig.6). Another post-processing option is to use a CAD tool, such as ANSYS SpaceClaim, to create a uniform lattice model (STL), and to use GSAM toolkit to map the lattice optimization result to a new variable thickness lattice model, as illustrated in Fig.7.

For more information
Diego Boscolo - EnginSoft
d.boscolo@enginsoft.com

ESTECO: 20 years of innovation in the engineering design process

Interview with Carlo Poloni, President of ESTECO

ESTECO, the Italian software house specialized in numerical optimization and simulation data management, is celebrating the twentieth anniversary since its foundation. Today, the company has 120 employees distributed across Italy, India and the United States, more than 300 international industrial customers, and 200 prestigious universities that use the software for educational and research purposes. ESTECO technology is designed to support companies across a large number of industrial sectors in the continuous process of improving their products.

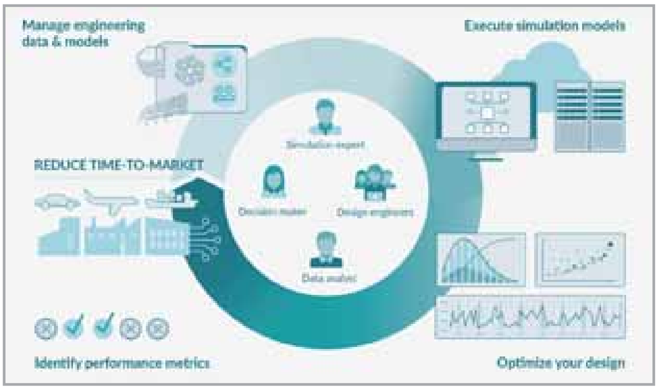
In this interview, Carlo Poloni, President of ESTECO, explains how their technology helps engineers to capture the simulation process, generate data from virtual models, interpret engineering data, and define the metrics for informed decisions, all with the goal of designing better products, faster.

Q: As virtual models become the norm in all stages of product design, and the benefits of simulation extend beyond domain experts, the amount of data in the hands of engineers has grown exponentially. How can we ensure an effective decision-making process in this fast-changing environment?

First and foremost, the models and measurements used to make decisions must be reliable. The same applies to the process that generates them. In most cases, too much information is equivalent to no information. Being able to filter, synthesize and validate generated or accessed data is the crucial part of any business process and has a strong impact on the consequent decision.

Engineering is no exception. In this case, however, the complexity increases since the data may be generated both from agnostic observations and from physics-based numerical models that have inherited half a century of research and validation activity.

Effective management of the enormous quantities of simulation data makes all the difference when it comes to turning that data



into actionable insights, which ultimately translate into more effective decisions earlier in the process.

Q: Evolving an idea into a product is a collaborative task requiring specialized contributions from different people and teams. How can ESTECO technology help break down the silos among engineering departments?

In the past, the dialog between teams was based on the satisfaction of requirements; the future will offer the opportunity to share predictive models capable of identifying what can be achieved and at what cost. There is a cultural (and organizational) shift that must be made: parametric models should be built upfront – and then maintained – to allow the subsequent design adaptations to be evaluated almost in real-time to leverage the highest competencies encapsulated in the predictive models.

Our simulation data management technology is already designed to enable real-time collaboration among cross-functional teams. It allows users to safely capture, version and share interdisciplinary engineering knowledge across the enterprise. Engineering teams can then successfully move towards the vision of the digital twin by incorporating simulation data and processes into their product development management system.

Q: The adoption of accepted standards for process representation is fundamental to guarantee interoperability between different systems. How does ESTECO technology respond to these challenges?

Products, by definition, have a multidisciplinary nature which means that each specialized area must represent its process in a way that others can understand and execute. Our technology supports the Functional Mock-up Interface (FMI) and Business Process Model and Notation (BPMN) standards to enhance integration and to facilitate enterprise-wide collaboration. This enables people to take advantage of software interoperability and connect with multiple tools. Standardization and integration are also key to make the engineering process a collaborative experience across the enterprise, enabling effective data management and sharing. We also want our technology to reach non-expert users, with the intention of extended the power of artificial intelligence and machine learning tools within the company.

Q: Artificial intelligence (AI) and machine learning are at the heart of your optimization-driven design approach. Can you indicate how this approach enables a dramatic reduction of time-to-market and development costs?

Traditional engineering design based on trial and error implies a time-consuming and costly method of manually conducting simulations until the optimal solution is found. Optimization technology to autonomously explore and select promising configurations is essential to limit the search for a design solution to a focused area of interest to maintain efficiency in the design process. Our 20 years' experience in numerical methods allows us to continuously deliver intelligent and autonomous algorithms to handle hundreds of design parameters simultaneously, balance complex tradeoffs, and quickly identify a set of optimal solutions, even for the most elaborate and constrained design problems.

Q: Talking about unlimited computing resources (Cloud-HPC) and, consequently, issues of data sharing and protection, what is your opinion about this current trend in the simulation market?

Nowadays, cloud computing makes almost unlimited computing resources available to users and the current data centers can adapt to the resources required. However, some issues related to software licenses and costs may practically limit the number of parallel instances being executed. Security and the system resilience to support large ecosystems are of key importance for efficient and effective decision making in the digital world. Engineering organizations need to be able to share knowledge in real-time, breaking down silos geographically and across departments, but they also need to protect their intellectual property with the highest security standards. Role-based tiered access to company knowledge within a secure web-based environment enables more efficient collaboration throughout the design process.

Q: ESTECO has grown steadily over the years, expanding its software offering and earning substantial recognition. How do you intend to keep growing for another 20 years?

When we founded ESTECO, multi-objective optimization was a virtually unknown topic: nonetheless, we persevered and pursued our vision ahead of the times. I believe that, along with a certain amount of luck, this was the key to our success. Even after 20 years, I continue to consider ESTECO a start-up: aside from our core business, we continue to learn and are always searching for new ways to innovate.

ESTECO is a sponsor of the International CAE Conference and Exhibition 2019 and is participating in the International CAE Conference with a presentation in the Energy Oil&Gas session

www.caeconference.com

Coupling CFD and particle modeling extends range of granular-fluid system analysis

ROCKY

By Lucilla Almeida
ESSS

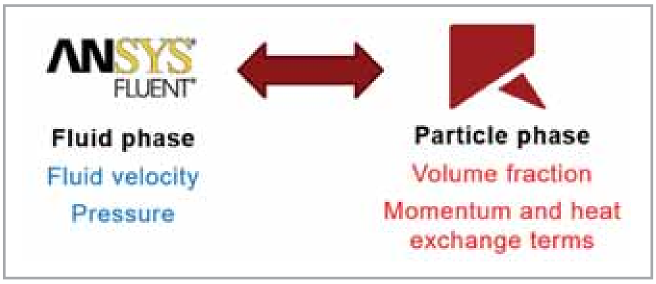


Fig. 1 - Two-way ANSYS Fluent-Rocky DEM coupling

Can you determine what these particle-related applications have in common?

- A sugar cane bagasse separation process that must withstand a variety of operating conditions
- A coating equipment application in which ring-shaped particles fall near the gas inlet, blocking gas flow and collapsing the bed
- A cyclonic vacuum cleaner designed specifically to pick up pet hair
- A pharmaceutical tablet-coating device that injects hot air to dry particles
- The erosion caused by sediment flow in an oil-and-gas-application pipe bend

Because each of these applications integrates fluid behaviors that affect particulates, they can all be successfully analyzed and optimized using a coupled computational fluid dynamics-discrete element modeling (CFD-DEM) solution.

Traditional methods alone cannot capture bulk particle behavior or fluid-particle interaction; a Multiphysics approach is required. ***ANSYS Fluent*** and ***Rocky DEM*** can easily be coupled to enlarge the range of granular-fluid systems that your R&D team can model – whether your market is industrial equipment, mining, or process equipment.

How ANSYS Fluent-Rocky DEM coupling works

There are two coupling modes:

- In a ***one-way*** approach, the fluid field affects the particle flow, but the particle flow doesn't, in turn, affect the fluid field. This method is particularly useful for simulating dilute flows.
- In a ***two-way*** approach, the fluid flow calculated in ANSYS Fluent affects the flow of particles in Rocky DEM, while Rocky-calculated particles change the flow of the fluids in ANSYS Fluent.

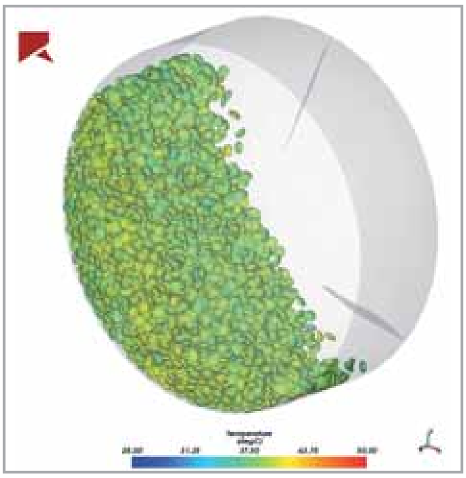


Fig. 2 - CFD-DEM pharmaceutical tablet-coating simulation using custom tablet shapes

In both approaches, all particles are tracked in a LaGrangian way by the DEM solver, explicitly solving the equations that govern translational and rotational particle motion along with the energy balance on the particle. These equations take into account the forces and torque on particles due to the fluid phase.

But in two-way coupling, given the pressure and fluid velocities computed by ANSYS Fluent, Rocky DEM computes the particulate phase volume fraction along with the momentum and energy exchanged between particles and fluid phases.

These terms are then transferred to the CFD solver, which uses this information to solve the equations that govern fluid flow, updating pressure and fluid velocities. This two-way exchange of information continues for each new time step until the full granular-fluid model is revealed (Fig. 1).

When to couple

The main advantage of the Fluent-Rocky coupled approach is that you can accurately solve cases in which particles have unique, non-spherical shapes (Fig. 2). Such simulation predicts real-world behavior, since particle-particle and particle-boundary interactions are solved, and all forces acting on particles are computed, on the DEM side.

Rocky's precise shape representation (including custom convex and concave shapes, flexible fibers, and shell particles) in combination with its several laws for computing fluid forces on particles increases model accuracy. Similarly, you can model adhesive/cohesive materials using one of the adhesion models available in Rocky.

In addition, since the DEM solver tracks each individual particle, there is a complete history for all particles inside the domain (such as velocities, temperatures, and contact data).

When combined with Rocky's extensive post-processing tools, the DEM software extracts a high level of information from the coupled Fluent-Rocky simulation, providing better insight into your application problem and ultimately reducing development costs and time to market.

For simulation cases that have a huge number of particles, Rocky's multi-GPU capabilities enable you to combine several GPU cards for the DEM solver with the Fluent distributed parallel option for solving the CFD equations.

There is one other way to broaden the range of cases that you can numerically model and expand your analyses to the next level. You can use specific Rocky features (adhesion/cohesion modeling, detailed particle collision statistics, energy spectra analysis) simultaneously with Fluent-Rocky coupling.

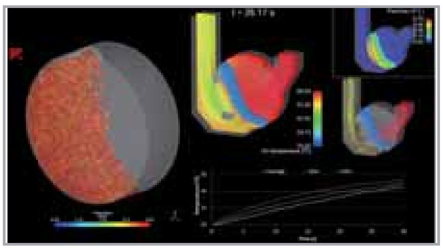
Real-world examples of ANSYS Fluent-Rocky DEM coupled simulations

In a coating equipment application, small spherical particles are fluidized due to the air flowing from the bottom while some "ring-"shaped particles are kept suspended due to the collisions of the small particles against the ring surface. A dense drag law is used to capture the fluidization behavior. Particles are colored in Rocky DEM by their initial position using the division tagging post-processing tool. This simulation helped engineers to predict scenarios in which the rings are not suspended and fall near the gas inlet, blocking the gas flow and collapsing the bed.

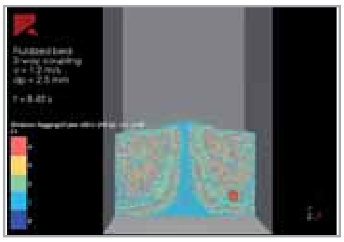
For this cyclonic vacuum cleaner designed to clean up pet hair, a large number of hair strands are modeled using Rocky DEM's flexible fiber particle shape and one-way Fluent coupling. A drag model adequate for long, slender particles was adopted to incorporate the effect of fluids on hair fibers. Using Fluent-Rocky coupling as a hair-modeling tool enables conceptual device testing, reducing the number of lab-tested prototypes and, therefore, minimizing development time and cost.

A large-scale pharmaceutical tablet-coating device is modeled using two-way Rocky-Fluent coupling and multiple domains with non-conformal and moving meshes. Custom particle shapes accurately predict tablet heating as hot air flows within the equipment, taking into account both convective and conductive heat transfer. The simulation

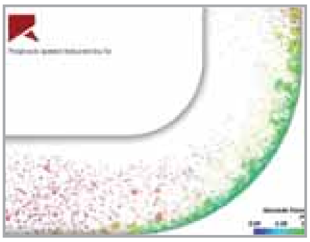
Flexible hair strands in a cyclonic vacuum cleaner. Simulation courtesy Bissell.



Pharmaceutical tablet-coating device showing the dry process of particles due to the injection of hot air



Rings suspended in fluidized bed of coating equipment



Pipe bend flow using real-particle-shape simulation to resolve erosion

allows engineers to predict the residence time and temperature distribution of particles for different operational conditions.

One-way CFD-DEM coupling can be a powerful tool to improve understanding of the sugarcane bagasse pneumatic separation process, whose product is sometimes used as biofuel for energy generators. Different particle shapes depict the pulpy residue, and air flow rate is varied to evaluate different operational conditions. Ganser drag law takes into account particle shape and alignment with flow; it also allows accurate prediction of separation efficiency.

In this final example, Rocky's wear model is used in a one-way coupling simulation with Fluent to predict surface modification due to erosion on a pipe bend, caused by sedimentary particle interaction with the surfaces.
Note that these coupled CFD-DEM simulations all provided the understanding necessary to optimize related equipment or processes.

More information is available at these additional links:

- Parametric and Multiphysics simulations (https://rocky.esss.co/library/webinar-parametric-and-multi-physics-simulations/)
- DEM and CFD Coupling (https://rocky.esss.co/library/webinar-dem-and-cfd-coupling/)

Lucilla Almeida, CAE Specialist at ESSS, holds BE and MSc degrees in chemical engineering and a PhD in nuclear engineering from the Federal University of Rio de Janeiro. At ESSS, she spent five years focusing on applying CFD to solve common engineering problems in the oil and gas industry, dealing with turbulent and multiphase flow simulations. Currently, she is an application engineer for Rocky DEM, supporting users, working on consultancy projects, and validating models implemented for the CFD–DEM coupling.

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

Rocky is a sponsor of the International CAE Conference and Exhibition 2019 and ESSS is participating in the International CAE Conference with a presentation in the Energy Oil &Gas session and another one in the Manufacturing session.

www.caeconference.com

Entry-level version Particleworks fluid simulation software released

Makes trial-and-error design studies for fluid phenomenon available to more design engineers

By Massimo Galbiati[1], Marisa Zanotti[1], and Akiko Kondoh[2]
1. EnginSoft - 2. Prometech Software, Inc.

Liquid flow simulation in design evaluation and for problem solving has long been one of the big challenges for experienced CAE engineers. A lot of effort is required to define the meshing and simulation models and a lot of time is required for calculation when using conventional grid-method computational fluid dynamics (CFD) software – even for simple and standard liquid flows. The evaluation of realistic phenomena that include such complicated liquid behaviors has often been thought almost impossible.

Particleworks takes a completely different approach, called moving particle simulation (MPS), to solve those problems and, since its first release about ten years ago, it has been introduced in many industrial fields because of its mesh-free, easy-to-use operation and unique simulation capabilities for liquid flow phenomena.

In early 2019, Particleworks for SOLIDWORKS, a newly developed entry-level version of the fluid simulation software Particleworks, was released as an add-on tool for SOLIDWORKS®. Particleworks for SOLIDWORKS is a frontloading CFD Software that allows the user to perform all operations in the SOLIDWORKS environment, allowing more design engineers to easily and effectively perform trial-and-error design studies. It includes functions for fluid phenomenon that are in high demand at design sites but that are difficult to reproduce with the conventional grid method, such as splashing, mixing and spraying.

Minimal input and simple operation to analyze results

The software facilitates designs for fluid phenomena during the planning and design stage by requiring minimum user operation. Firstly, the user creates a simulation in SOLIDWORKS (fluid area, polygon wall, inflow port, probe area) and defines the parts' movements. Next, clicking on the Particleworks for SOLIDWORKS icon, reveals menus in which the user defines the simulation area and the target parts, the material property values and inlets, the simulation parameters and the job environment. Once the calculation is executed, the simulation results can output to a color map display, a stream line display, a particle flow trace display, a cross section display, an animation and more, to be reviewed in SOLIDWORKS.

Typical fluid applications

Particleworks for SOLIDWORKS covers in-demand analysis areas such as splashing, mixing and spraying and can be used in various fields such as for tank oil sloshing, gear oil lubrication, the mixing of medicines, or foods, the emulsification of cosmetics, nozzle jets. Particleworks capabilities integrated into 3D-CAD allows users to immediately understand the influence of liquid behaviors caused by design changes. For instance, Fig. 1 shows a comparison among different design patterns for sprinklers.

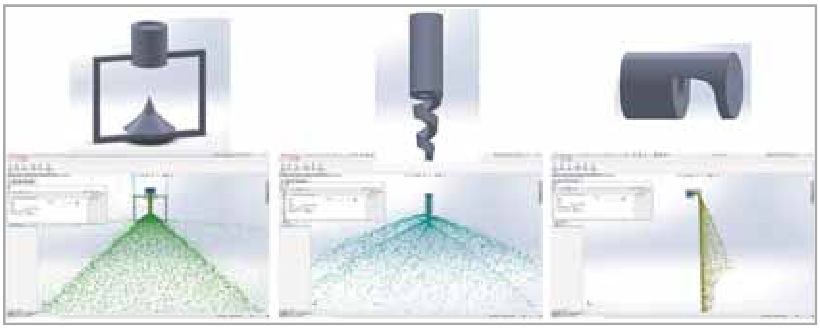


Fig. 1 - Comparison of spraying behaviors among different nozzle designs

For more information
Massimo Galbiati - EnginSoft
m.galbiati@enginsoft.com

ParticleWorks Europe launched after JV between Prometech Software and EnginSoft

Prometech Software Inc. and EnginSoft S.p.A. have announced the formation of a joint venture company, to be known as Particleworks Europe s.r.l., incorporated in Italy. The JV aims to establish, promote and manage the European distribution network of Prometech – software technologies and applications based on the Moving Particle Simulation (MPS) method and Multi-GPU acceleration which are currently known in the market under the commercial labels Particleworks and Granuleworks. In addition to and to supporting and enhancing this network, Particleworks Europe will also more specifically constitute the European competence center for these technologies, both to support distributors and to assist customers who require help for complex applications, verticalization, dedicated developments or integration into their scientific information systems. Particleworks Europe will also aggregate the needs and requirements of target customers to ensure their rational inclusion in the development roadmap of the related software technologies.

"I am honored by the establishment of Particleworks Europe as a joint venture with EnginSoft, the leading engineering simulation company in Europe." says Toshimitsu Fujisawa, Founder and CEO of Prometech Software Inc. *"By combining the strengths of both companies, that is, Prometech with its cutting-edge simulation technology, and EnginSoft with its excellent experience in SBES, the joint venture will provide attentive and proactive supports for users to utilize Particleworks more effectively. Prometech and EnginSoft will strengthen their strategic partnerships, and jointly continue efforts to improve simulation products and service in order for more users to utilize simulation technology in a more valuable way."*

Tsuyoshi Sumiya, COO of Prometech Software, Inc. states, *"Particleworks has been used by many domestic companies in the automotive industry. In Europe, we have been providing technical and marketing support through EnginSoft and other distributors. We are confident that we will be able to provide customers from a closer standpoint and with the same quality support as that of Japan thanks to the establishment of a European headquarters."*

"Particleworks Europe is the most representative and potentially effective expression of the EnginSoft business model, designed to transfer knowledge of specialist or vertical technologies for engineering simulation in the highly fragmented territory (in terms of countries, languages and cultures) that is Europe," says Stefano Odorizzi, founder and CEO of EnginSoft. *"Based on a JV with the technology manufacturer, and taking advantage both of EnginSoft's knowledge gained over 40 years in the market and the group's experience in networking, the EnginSoft model ensures both a widespread presence on the ground, and timely technical support, high professionalism, and the ability to ensure the effective inclusion of technologies in the design processes of customers,"* he continues.

Massimo Galbiati, co-founder and CEO of Particleworks Europe, comments, *"The JV will benefit from the mature collaboration between the two companies, Prometech and EnginSoft which was constructed during the pioneering phase of promoting Particleworks in Europe, and then confirmed by success well beyond the expectations, both in terms of installations and customer satisfaction. The marketing system to be executed by the JV will boost the initiative's success far beyond the current excellent results."*

The Particleworks Europe team will be present at the International CAE Conference from 28-29 October 2019 where they will be formally announcing the new joint-venture company.

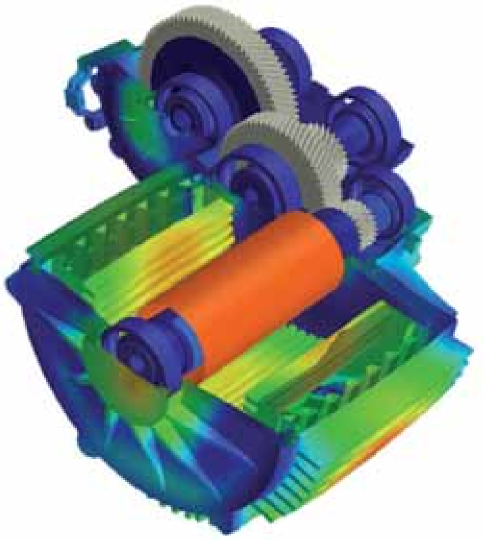
About Prometech Software, Inc.

Prometech Software, Inc. is a start-up company founded in 2004 that originated from the University of Tokyo. Prometech maintains a close relationship with the University and many other academic and non-academic organizations focusing on research and the application of the latest research to the manufacturing industry. Innovative simulation software products such as Particleworks that is based on the world's first commercial particle method fluid simulation software released in 2006, and Granuleworks (granular material simulation software) released in 2017, and the technology which extends from physical model development to HPC's and Computer Graphics are used in a wide range of industries, including automobiles, machinery, electrical machinery, chemistry, food, medicine, civil engineering, disaster prevention, and energy.

Romax Technology signs distribution agreement with EnginSoft to expand reach into Italy

EnginSoft SpA, global consulting company in the field of Simulation Based Engineering Science (SBES), has signed a distribution agreement for Italy with Romax Technology Limited, provider of world-leading simulation software and global services for design, performance simulation and manufacturing of geared and electro-mechanical systems.

The agreement will allow EnginSoft to distribute Romax Technology's complete range of products and services in Italy and reinforces the company's commitment to provide continuous, flexible and accessible leading products and unique service solutions that maximize value for its customers.



Romax Technology has just launched Romax Nexus and is leading the industry changing the world with its introduction of the most complete and customizable solution for designing mechanical and electro-mechanical powertrains. It is a powerful, integrated eco-system that allows each product to seamlessly communicate with other products both within and outside of Romax Nexus, to satisfy users in many different roles, departments and industries.

Users of the software can choose between investigating single specific aspects, or setting up broad, comprehensive studies. The suite's functionalities include concept design, structural analysis, durability analysis, noise, vibration and harshness (NVH) assessment, efficiency analysis, and more. All the products and options available leverage the knowledge that Romax Technology has gained over years of research and collaboration with the largest companies worldwide.

"As a leading global engineering solution provider, we're always seeking ways to create valuable partnerships with software vendors and to extend our portfolio in different markets," said Stefano Odorizzi, Chief Executive Officer at EnginSoft. He added, "With Romax Technology's innovative simulation and consultancy we are becoming more aggressive in a market sector where we see great potential for business growth, also resulting from the upcoming transition to electric vehicles which will generate a huge demand for more compact, more efficient, more silent, and better performing transmission systems."

"We at Romax Technology are very excited to partner with EnginSoft the Italian distributor of our solutions. The EnginSoft brand's great reputation in the field of Simulation Based Engineering Science, in combination with its staff's proven competence in, enthusiasm for, and knowledge of the Italian market, represent the perfect mix to increase our market share in Italy," Andy Poon, CEO at Romax Technology, commented.

He added, "We provide the most advanced software solutions and combined engineering expertise to support the entire design and development cycle of geared and electro-mechanical drive systems. Our latest partnership brings us closer to our customers than ever before, working with a reliable local partner that knows the network of companies throughout the territory. The collaboration between EnginSoft and Romax Technology leverages the companies' respective strengths."

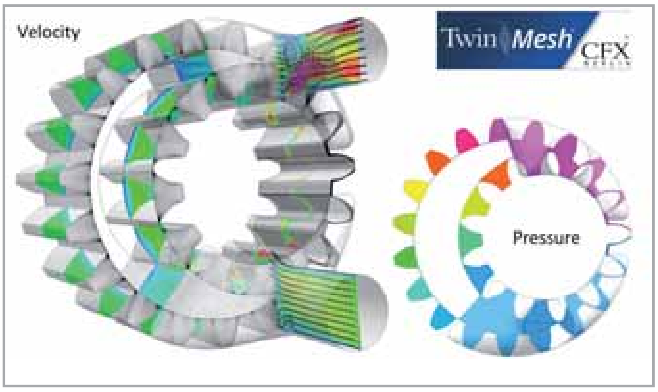
For more information on Romax Technology, visit www.romaxtech.com

Romax Technology is a sponsor of the International CAE Conference and Exhibition 2019

www.caeconference.com

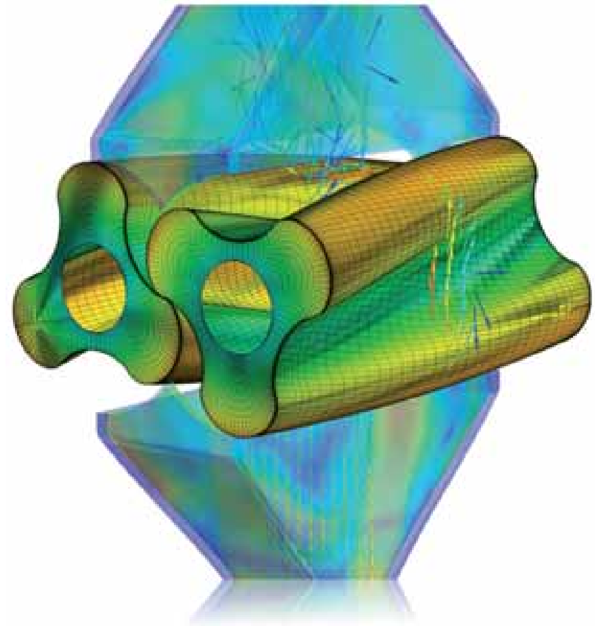
First International TwinMesh Users' Summit announced

Scheduled at 2019 International CAE Conference



CFX Berlin and EnginSoft have announced that the 2019 International CAE Conference in Vicenza in October will host the first International TwinMesh Users Summit, as an Exhibitor's Initiative.

The International TwinMesh Users' Summit is planned both for existing TwinMesh users as well as for engineers interested in reliable CFD simulation for rotary positive displacement (PD) machines. Complementary to ANSYS CFD, TwinMesh is a grid generator which



automatically generates high quality hexahedral grids for the moving parts of rotary PD machines such as gear and gerotor pumps, screw and scroll compressors, as well as lobe pumps. When combined with ANSYS CFD, the TwinMesh approach allows highly efficient, reliable CFD analysis of these machines thereby helping engineers to understand and improve their designs.

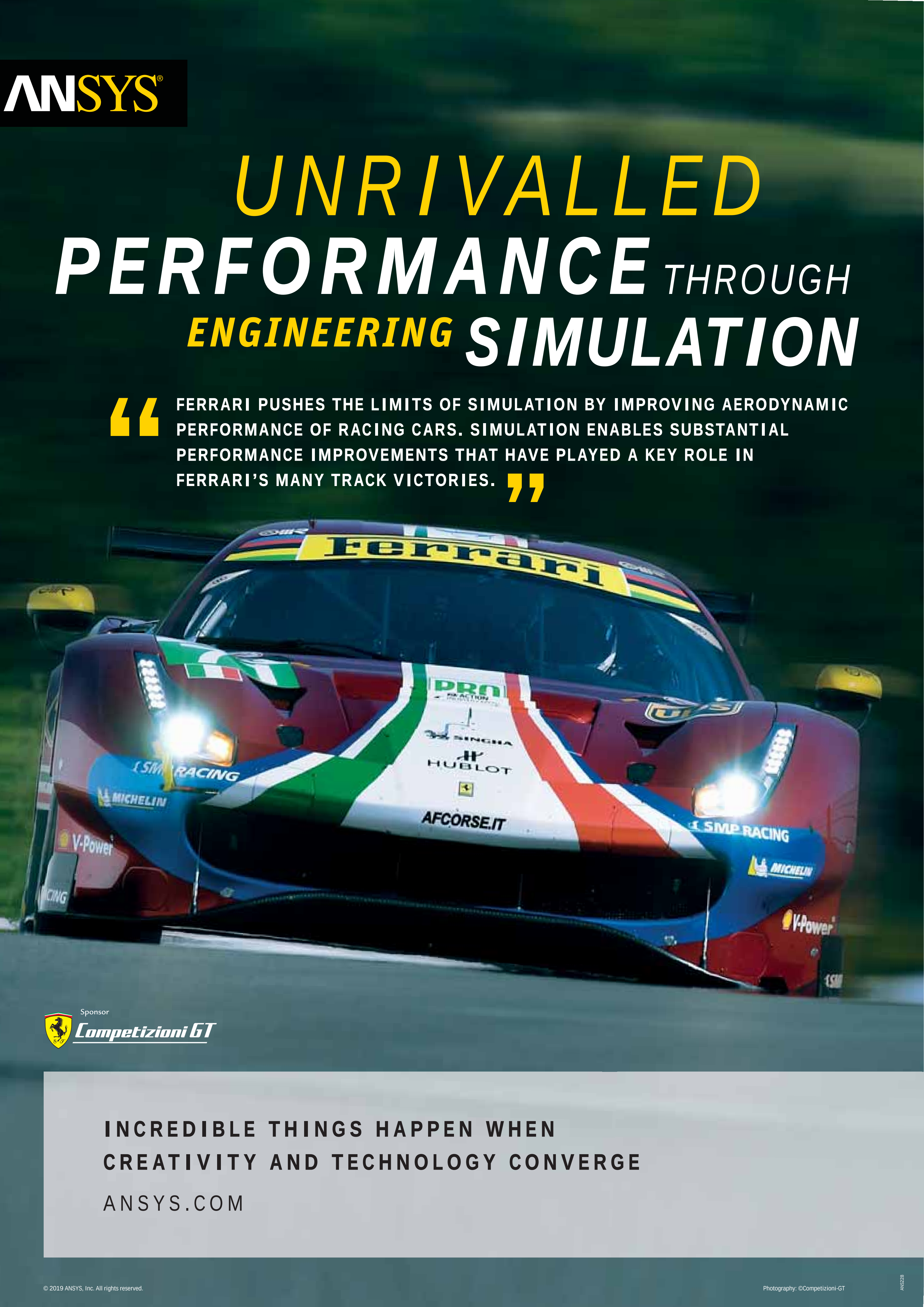
The users' summit will take place on 28 and 29 October at the TwinMesh exhibition booth and will offer several live sessions for visitors to learn about the easy, efficient TwinMesh workflow, best-practice guidelines, interesting application showcases, new features in the current and upcoming TwinMesh product releases, as well as first-hand user experience.

International CAE Conference participants can find the latest timetable for the TwinMesh live sessions at the TwinMesh booth, where they will also be able to book an individual meeting with the TwinMesh experts. Participants can attend either specific sessions or the whole summit.

For more information:
Alessandro Arcidiacono - EnginSoft
a.arcidiacono@enginsoft.com

TwinMesh is a sponsor of the International CAE Conference and Exhibition 2019 and CFX Berlin is participating in the International CAE Conference with two presentations in the Energy Oil&Gas session.

www.caeconference.com


ANSYS
UNRIVALLED
PERFORMANCE THROUGH
ENGINEERING SIMULATION
"FERRARI PUSHES THE LIMITS OF SIMULATION BY IMPROVING AERODYNAMIC PERFORMANCE OF RACING CARS. SIMULATION ENABLES SUBSTANTIAL PERFORMANCE IMPROVEMENTS THAT HAVE PLAYED A KEY ROLE IN FERRARI'S MANY TRACK VICTORIES."
Sponsor
Competizioni GT
INCREDIBLE THINGS HAPPEN WHEN
CREATIVITY AND TECHNOLOGY CONVERGE
ANSYS.COM
© 2019 ANSYS, Inc. All rights reserved.
Photography: ©Competizioni-GT