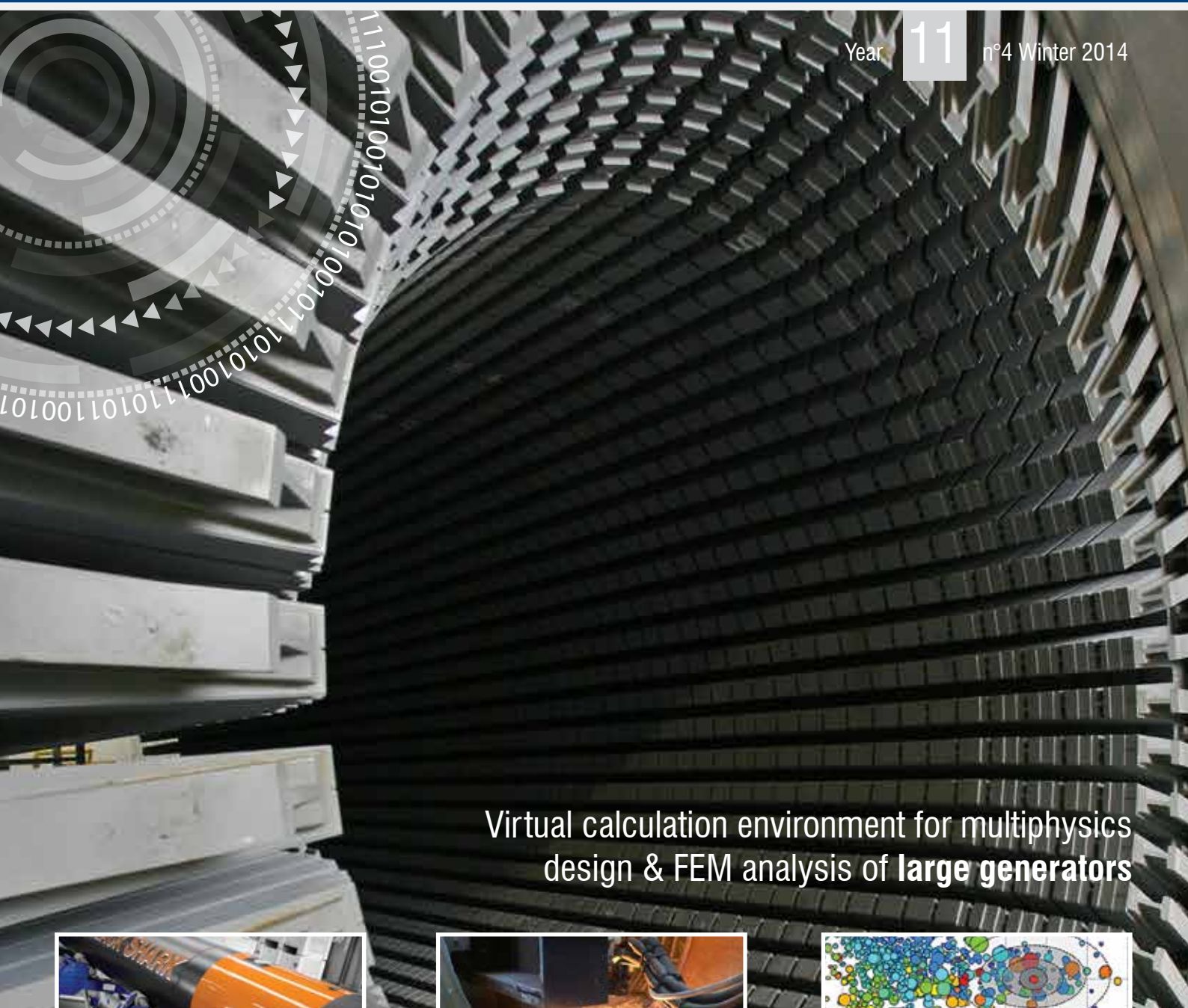




# Newsletter

Simulation Based Engineering & Sciences

Year **11** n°4 Winter 2014



Virtual calculation environment for multiphysics design & FEM analysis of **large generators**



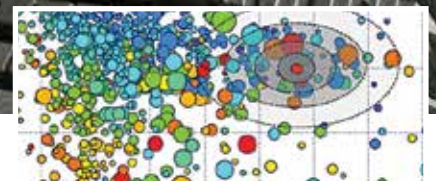
**WASS:** a history of Experience in the FEM World

New 1D simulation developments to meet current and next car generation **engine cooling challenges**



**SIMIC:** an Italian company among the protagonists of the ambitious **ITER project**

Residual stress of **railway wheels:** a predictive method



Thermo-hydraulic simulations in STX France **ship networks**

Optimization of a low noise **hydraulic piston pump** to minimize the noise emission



*Tecnologie per il Calcolo Numerico*

## CONSORZIO TCN

Technologies for Numerical Computation  
ADVANCE TRAINING CENTER

*High quality and the most up-to-date competencies play an integral part in business competitiveness and job advancements*

TCN Group offers High Level Training for Engineers.

For 2015, you can expect:

a large **catalogue of courses**, rich in training applications for the diffusion of those disciplines pertinent to numerical simulation; **customized trainings**, created from the requirements of the company, **continuous Training** aimed to increase competencies, both in business and professional contexts, available thanks to the support of funds from the Institute for the Development of Vocational Training of Workers; a dedicated portal for **distance engineering training**

Among the catalogue of course subjects are:

- Fracture mechanics
- Acoustics
- Fatigue Design
- Cases relative to friction and wear
- Mechatronics
- Impact Dynamics
- Data mining
- FE Dynamic Analysis
- CFD
- Experimental Data acquisition systems
- Composite Materials Design
- Multibody methods for kinematic and dynamic design of mechanical systems
- Design of experiments
- Project management

# FLASH

*"I think there is a world market for about five computers"* - Thomas J. Watson, former chairman and CEO of IBM, reported to have said this in 1943.

Fortunately, he was wrong. From today's perspective, computers are more important than ever before - particularly in Engineering. Just think of the variety of CAE Technologies used every day via portable personal machines and computers; they have revolutionized the world of engineering. The time-to-market of new products and scrap is reduced, production resources are used more efficiently and designers are equipped with tools to develop innovative new products.


In this edition of our quarterly Newsletter, we highlight CFD in many of its facets. We examine CFD approaches to avoid the dangerous ice accretion in high altitude aircraft using FENSAP-ICE from NTI. It is possible to understand the effect this phenomenon has on engine damage and power loss - thereby avoiding life critical accidents. Once the ice accretion simulation is performed, it is possible to assess the performance loss due to icing, and analyse if the engine still provides an acceptable thrust to the aircraft.

EnginSoft are proud to have been a member of the FIRST project, which provided an understanding of the fundamental importance of complex physics for spray atomization, using CFD software, such as ANSYS CFX. Read on to get an in-depth understanding on the project achievements along with many more cases in this Newsletter.

Many of the success stories were also presented at the latest International CAE Conference - which celebrated 30 years of innovative success for EnginSoft. This year, thanks to the commitment of our experts, partners and sponsors the Conferences' attendees' experienced richer topics and contents than ever before. Again, it is evident that EnginSoft is a point of reference in a growing network, formed by numerous relationships with highly qualified industrial members, as well as remarkable members from leading global Universities, Research Centres and High-Tech Companies.

Moreover, we like to promote the future of engineering through our CAE International Poster Award. The best technical posters submitted by students and researchers in the field of CAE were awarded in order to encourage and disseminate the use and application of simulation tools in the academic field. EnginSoft remains a driving force in shaping the engineering future, which would not have been possible without our clients and partners. Thus, I would like to take the opportunity to thank all of you for your contribution to the overwhelming success of last year's various research and consulting projects. In addition, a great thank you to all participants and supporters of the International CAE Conference 2014, for making this event a remarkable success for global engineering. We are looking forward to continuing our engineering partnership with you in the years ahead. EnginSoft wishes you, your colleagues and families a very happy and prosperous New Year.



Stefano Odorizzi, Editor in chief  




# Contents

## INTERVIEW

- 6 SIMIC: an Italian company among the protagonists of the ambitious **ITER project**

## CASE HISTORIES

- 8 **WASS**: a history of Experience in the FEM World
- 10 New 1D simulation developments to meet current and next car generation **engine cooling challenges**
- 14 Residual stress of **railway wheels**
- 17 Flowmaster used for **thermo-hydraulic flow simulations** in STX France ship networks
- 16 Development of a virtual calculation environment for multiphysics design & FEM analysis of **large generators** with particular focus on the electromagnetic analyses executed by means of proprietary algorithms embedded within ANSYS Maxwell
- 22 Flowmaster used for thermo-hydraulic flow simulations in STX France **ship networks**
- 24 Piping afloat: Using CAE to calculate **pipng systems onboard ships**
- 26 Design of the **Roller Coaster LOOP 395**
- 28 Topological Optimization methods for **aerospace applications**, weight reduction of a RTM Composite RIB
- 30 Optimization of a low noise **hydraulic piston pump**
- 33 ANSYS Mechanical **customization** through ACT extensions
- 36 Topological Optimization procedure for **civil steel frame design**
- 39 Computational Fluid Dynamics prevents **engine damage in Ice Crystals** conditions
- 40 Multi-Objective Optimization Analysis for

- Selection of Eco-Friendly Materials According to Itaca Protocol for **Zero Energy Buildings** in the Mediterranean Climate
- 44 Design and modeling the  $\phi$ -F chart method for **active solar energy systems** in Scilab
- 46 **Simulation governance**  
An idea whose time has come

## SOFTWARE UPDATE

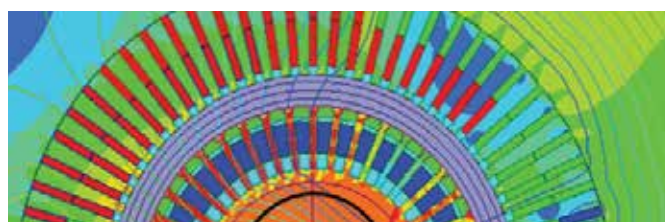
- 48 Easy Geometry Preparation in ANSYS **SpaceClaim** Direct Modeler
- 50 Simulation of combustion system **Reaction Design's Products**

## RESEARCH

- 53 The Intelligent Preventative **Healthcare Solution**
- 54 An overview of the **FIRST project**: CFD can be a valuable tool for a deeper understanding of the atomization process

## EVENTS

- 57 The meaning of being partner
- 58 TWO-DAY intensive workshop booked for ANSYS Finmeccanica users
- 60 INTERNATIONAL CAE CONFERENCE 1984 – 2014: thirty years ago, the FUTURE begins here ...
- 62 International CAE Poster Award 2014: a showcase for young researches in love with innovation



# OUR ACKNOWLEDGEMENT AND THANKS TO ALL THE COMPANIES, UNIVERSITIES AND RESEARCH CENTRES THAT HAVE CONTRIBUTED TO THIS ISSUE OF OUR NEWSLETTER



## Newsletter EnginSoft Year 11 n°4 - Winter 2014

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

*All pictures are protected by copyright. Any reproduction of these pictures in any media and by any means is forbidden unless written authorization by EnginSoft has been obtained beforehand. ©Copyright EnginSoft Newsletter.*

### EnginSoft S.p.A.

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

[www.enginsoft.it](http://www.enginsoft.it) - [www.enginsoft.com](http://www.enginsoft.com)  
e-mail: [info@enginsoft.it](mailto:info@enginsoft.it)

*The EnginSoft Newsletter is a quarterly magazine  
published by EnginSoft SpA*

## COMPANY INTERESTS

CONSORZIO TCN [www.consorziotcn.it](http://www.consorziotcn.it) • [www.improve.it](http://www.improve.it)

EnginSoft GmbH - Germany  
EnginSoft UK - United Kingdom  
EnginSoft France - France  
EnginSoft Nordic - Sweden  
[www.enginsoft.com](http://www.enginsoft.com)

Cascade Technologies [www.cascadetechnologies.com](http://www.cascadetechnologies.com)  
Reactive Search [www.reactive-search.com](http://www.reactive-search.com)  
SimNumerica [www.simnumerica.it](http://www.simnumerica.it)  
M3E Mathematical Methods and Models for Engineering [www.m3eweb.it](http://www.m3eweb.it)

## ASSOCIATION INTERESTS

NAFEMS International [www.nafems.it](http://www.nafems.it) • [www.nafems.org](http://www.nafems.org)  
TechNet Alliance [www.technet-alliance.com](http://www.technet-alliance.com)

## ADVERTISEMENT

For advertising opportunities, please contact our Marketing office at: [newsletter@enginsoft.it](mailto:newsletter@enginsoft.it)

## RESPONSIBLE DIRECTOR

Stefano Odorizzi - [newsletter@enginsoft.it](mailto:newsletter@enginsoft.it)

## ART DIRECTOR

Luisa Cunico - [newsletter@enginsoft.it](mailto:newsletter@enginsoft.it)

**PRINTING** Grafiche Dal Piaz - Trento

The EnginSoft Newsletter editions contain references to the following products which are trademarks or registered trademarks of their respective owners: ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. [ICEM CFD is a trademark used by ANSYS, Inc. under license]. ([www.ANSYS.com](http://www.ANSYS.com)) modeFRONTIER is a trademark of ESTECO Spa ([www.esteco.com](http://www.esteco.com)) - Flowmaster is a registered trademark of Mentor Graphics in the USA ([www.flowmaster.com](http://www.flowmaster.com)) MAGMASOFT is a trademark of MAGMA GmbH ([www.magma-soft.de](http://www.magma-soft.de)) - FORGE, COLDFORM and FORGE Nxt are trademarks of Transvalor S.A. ([www.transvalor.com](http://www.transvalor.com)) FENSAP-ICE is a trademark of NTI Newmerical ([www.newmerical.com](http://www.newmerical.com)) - LS-DYNA is a trademark of LSTC ([www.lstc.com](http://www.lstc.com)) - Particleworks is a trademark of Prometech Software Inc. ([www.prometech.co.jp](http://www.prometech.co.jp))



# SIMIC: an Italian company among the protagonists of the ambitious ITER project

ITER is a large-scale scientific experiment that aims to demonstrate the technological and scientific feasibility of fusion energy

The Company SIMIC, established in 1975, was initially oriented to serve the market of light carpentry and industrial plant installations & maintenance.

In 1981 the small company took a new form, specializing in the field of the heavy carpentry and began to focus on the manufacturing of pressure equipment and heat exchangers, while the installation activity grew in parallel.

From 1987 SIMIC continues to develop their well-equipped machining department, today endowed with numerical control machinery for the manufacturing of large sized pieces.

Since 1999, SIMIC plays an outstanding role in the supply of cryostats and vacuum equipment for an elite group of international companies and organizations, leading parts of today's scientific research, SIMIC has been at the cutting edge of Europe's projects in this field.

For nearly three decades SIMIC has fed its constant growth by means of continuous investments in new technologies and dynamic and specialized personnel that, together with a tightly structured and flexible organization, constituted its true core value and strongly supports the company.

We have interviewed Dr. Marianna Ginola, Sales Director of SIMIC

**1. ITER Project will be funded by major investments engaged with Italian companies whom have proved to be capable to take on this technological challenge in order to overcome 60% of the total value of this European order. In Italy there is a common understanding that believes the**





**Italian productive system tends to favor small and medium enterprises over clusters in order to compete on large global projects. What was your view on this and was this the case for the ITER Project?**

As far as the ITER project is concerned, obtaining the most relevant contracts for Italian companies has been possible thanks to the cluster of companies aiming at gathering their different competences to meet the very specific technical, managerial and financial requirements of the project.

It's well known that the Italian productive system is mainly constituted by small and medium enterprises with a very limited attitude towards clustering, but in this case we have successfully overcome this limitation.

Nevertheless, it is not always true that gathering companies together is the best way to work and sometimes small and medium enterprises find a better solution in subcontracting. I think that in Italy we still have a lot of possibilities for this kind of contracts and that the best type of collaboration has to be evaluated in order to provide the best match with the project challenges.

**2. What are your expectations, in terms of results, considering the synergic work of the Industry, Research Institutions and Public Administration in terms of best practice and technology transfer, as well as specific technical training and expertise are concerned?**

As far as cutting edge scientific and technological projects are concerned, like ITER, the synergy between industries and research institutions is fundamental, since it creates a positive loop in which the benefits are several, including technological development, process development, and training of human resources that are oriented in a rich technical growth. At the same time the experience of managing complex projects positively affects the companies and their management competence.

**3. SIMIC has won against the international competition and will build ITER super-conductive coils, which are essential to create high intensity magnetic fields. What are the technological attributes that distinguish your company from the national and international offering?**

I believe that our proposal has been selected considering different relevant aspects: the high technological content and good competitiveness, related to adequate plants, as the one in Porto Marghera which is characterized by extremely high capacity, direct access to the sea and a gantry-type boring machine on work that is one of the biggest in Europe. A very important role has also been played by the international collaboration we have set up in order to provide and accomplish the technical requirements, and in this specific case subcontracting has allowed us to keep project costs under control.

**4. What is the SIMIC attitude towards human resources development?**

Human resources represent the great value of our company and thanks to them, to their engagement and passion we could reach very important results. Our target is to grow our personnel as much as possible limiting the employment of new resources maintaining the flexibility that has always characterized our way of working.

**5 Which are your expectations in the recently established collaboration between SIMIC and EnginSoft?**

First of all, it's a great satisfaction having this contract assigned in Italy, allowing other Italian suppliers to give their contribution to ITER project. The technical and managerial requirements are very high and will demand a great engagement, attention and constant work to meet the contract objectives. The collaboration with EnginSoft is very promising and we are sure that we will find the professional expertise and the flexibility which are necessary to face this project in a successful way.





# WASS: a history of Experience in the FEM World

WASS (Whitehead Sistemi Subacquei) is named after the torpedo's inventor Robert Whitehead. WASS designs and manufactures defense underwater systems, from the first conceptual sketch to the last after sale customer support. WASS joined the Finmeccanica Group in 1995 and it now belongs to the Defense Division. Its foundation dates back to the second half of the 19th century. Its history is long and complex. It was first located in Fiume, then it moved to Livorno and has plants also in La Spezia and Napoli, not to mention its presence at some Italian Navy bases.

After more than one century, WASS is still at the top.

Many Navies all over the world have bought WASS systems throughout the course of history. WASS is present in the international market, in a great number of countries.

Every effort is being made to satisfy the market demand. WASS offers its customers a wide choice of product lines. The core business is represented by torpedoes, both heavyweight (HW) and lightweight (LW) ones. The Black Shark and the MU-90, to name but a few, have a huge success. WASS heavyweight torpedoes are wire-guided and can be launched from submarines or surface platforms, while WASS lightweight torpedoes can be launched from surface platforms, helicopters and airplanes. WASS countermeasures are launchable from submarines or surface platforms: they may jam the enemy's torpedo sensors or decoy the enemy far from the launching platform. WASS sonar systems are installed aboard the ship's bulb to detect potential threats.

In recent years, WASS has entered the civil market by developing underwater unmanned vehicles, such as the V-Fides UAV which is a R&D project.

WASS has a long tradition of designing torpedoes, but it has to look ahead if it wants to survive the aggressive competition in the market. Reducing time-to-market is the key to success. Nowadays, computers are much more powerful and Finite Element codes are much more versatile. Engineers just need to use them in a clever,



smart way. ANSYS products are the state-of-the-art in this field. WASS engineers started using them many years ago and continue to use them in an advanced way.

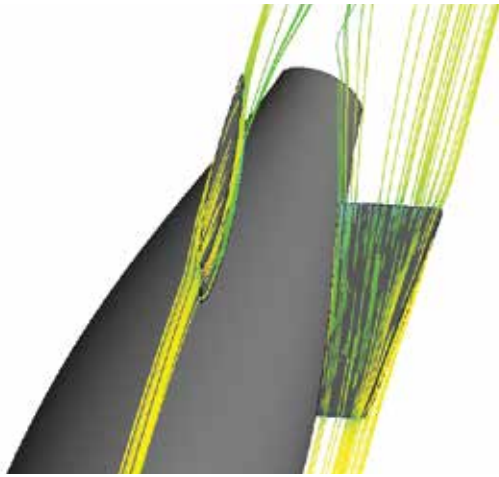
FEM is particularly convenient for WASS because that saves time and money.

The typical tests performed in WASS are pressure tests, shock and impact tests, climatic tests, pool tests and sea trials. Experimental tests, especially sea trials, are very expensive and in most cases they take a great deal of time. WASS cannot eliminate tests, because they are necessary in order to verify FEM models, to complete the qualification process, and so on, however the number of tests can be considerably reduced. Building the FEM model requires the major effort for engineers. Once done, engineers can analyze several different scenarios very quickly by changing the analysis conditions. This is the reason why WASS invests money and people in FEM.

WASS engineers take advantage of FEM tools on a daily basis.

From a mechanical point of view, both static and dynamic analyses are performed. Given that WASS systems operate underwater, buckling analyses are often conducted to investigate how torpedoes behave under external pressure to check performance in operational conditions. Most of the analyses performed in WASS





CFD streamlines on torpedo rudders



FE Torpedo impact on a target

are non-linear due to material and contact non-linearities. In regard to dynamic analyses, not only ANSYS is used, as LS-DYNA is a truly reliable explicit code that can efficiently study fast dynamic phenomena such as explosions and shocks. The ALE (Arbitrary Lagrangian-Eulerian) formulation is used to study the FSI (Fluid-Structure Interaction) between torpedoes and sea water, i.e. at water impact after flight if launched from surface vessel or aircraft. Steady state and transient thermal analyses are performed to predict the temperature distribution, for example on apparatus installed aboard a submarine or on electronic components mounted on PC boards inside a torpedo, where significant heat sources are present. CFD analyses are performed to predict the trajectory of systems moving underwater and the hydrodynamic forces acting on them. On some occasions, coupled Thermo-Fluid-dynamic analyses are required. The mesh of FE models built in WASS is predominantly hexahedral and mapped. Comparative analyses are often performed for example by varying the shell thickness, the shape of the stiffening ribs, etc. in order to minimize the weight or to verify different solutions.

Buckling analyses predict the critical sea depth and the typical buckling mode shapes (the torpedo shell deforms just as a Coke can, when the external pressure reaches its buckling critical limit). Local holes in the shell are checked because on the one hand, they may “trigger” and induce the global buckling of the structure, and on the other hand, they may suffer from stress concentration.

As far as heavyweight torpedoes are concerned, there are two types of target: submarines or ships. The torpedo explodes when it hits the submarine hull or at a distance below the ship keel: its effectiveness depends on its capacity to cause damage through leaks on submarines or to break ships. The effectiveness of the explosive train is often verified by performing LS-DYNA fast dynamic analyses.

Before carrying out a pool test on a single section or subsystem of a torpedo/countermeasure, LS-DYNA FSI water impact analyses are performed in order to find the pool test conditions equivalent to the real shock conditions in real seawater entry.

Whenever a new internal component has to be selected (i.e. a new type of shock absorbers to be mounted on an electronic rack located in the control section of a torpedo), ANSYS transient

dynamic analyses are very useful to verify it before installing it and testing the assembly. This sort of analysis may be used to identify the equivalent half-sine to be applied in a shock test, for example. From a hydrodynamic point of view, CFD analyses are important in order to limit the number of tests (hydrodynamic/cavitation tunnels and towing tanks are not widely available in the world and they are extremely expensive). ANSYS CFX is the code used by WASS to verify the external profile of torpedoes, to predict the exit of torpedoes from the launching tubes and their motion underwater, to estimate the hydrodynamic forces acting on systems, to optimize the geometry of empennages and propellers, etc. For complex CFD analyses (i.e. torpedoes launched from a moving submarine), the so-called mesh motion capability is used. At a pre-processing stage, ICEM is often used as a mesh generation software. FEM analyses are essential for WASS to be competitive in the international market.

*Giulia Filucchi*

*Mechanical Department, Finmeccanica-WASS*

Ask the expert in the defense sector simulation:

Roberto Gonella, EnginSoft

[r.gonella@enginsoft.it](mailto:r.gonella@enginsoft.it)



V-Fides UAV



# New 1D simulation developments to meet current and next car generation engine cooling challenges

Current car generation offers increased comfort and capabilities, but customer expectations about fuel economy and environmental concerns drive automotive industry to improve over and over. During the last decade thermal engineers have been focusing their attention on engine cooling systems, introducing new parts, new functions and devoting more resources to proving ground or real world validation sessions. As a consequence, vehicle level thermal, hydraulic and aerodynamic simulations have to evolve alongside improved testing.

This paper explores a possible approach to model current and nearly future car engine cooling systems based upon Flowmaster code, transient runs but basic vehicle and engine usage estimation. Final goal is getting a quick but reliable method for system sizing, alternate components checks, assessment according different environmental conditions or modified driving patterns and vehicle usage.

## Old dogs, new tricks

Internal combustion engines generate heat to deliver mechanical power but excess heat has to be removed to prevent failures. Nowadays, nearly all passenger cars are equipped with liquid based cooling systems, whose basic design is simple and efficient yet one century old: a water and antifreeze mixture removes heat from the engine and delivers it to a “radiator” that discharges it into the environment. Keeping the powertrain cool is the primary function of car cooling systems and driving reliable cars was fair

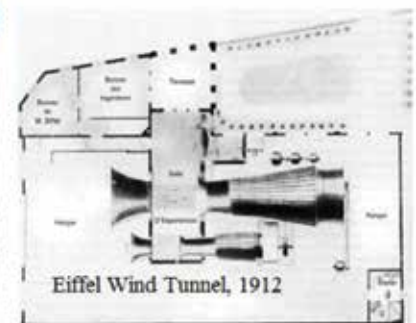


Fig. 1 - Most of terrestrial vehicle engine cooling system components and test facility technologies were available at the beginning of 20th century, but modern vehicle capabilities require better parts and methods (image authors: Atelier Mempli, Fong )



enough for our grand-grand fathers and mothers: but the larger the car market, the more demanding the customers so in order to increase passenger thermal comfort both a coolant pump and a thermostatic valve sensitive to coolant temperature were introduced. The pump delivers hot coolant whatever temperature or engine load or revs, the valve blocks cold coolant flow to the radiator, transforming the engine cooling system into an heat source to keep warm passengers cabin or defrost windscreen: a real safety improvement in cold weathers.

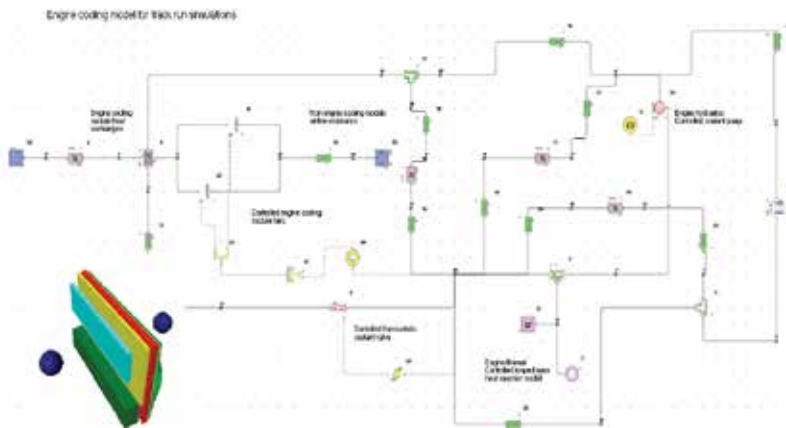


Fig. 2 - A Flowmaster 1D thermohydraulic simulation model built by KISS approach to investigate dynamic engine cooling system performances includes radiator module layout



Fig. 3 - All these vehicle share the same baseline engine, but their lap times and thermal results on track would be very different. Fia Formula F4 courtesy of Tatuus Race Car

The secondary function of engine cooling system as a thermal power sink raised interest of car makers when more challenging vehicle pollutant emission factors were imposed. If you can prevent coolant removing heat from the a cold engine or the gear box for a while after cranking up you can make engine metal and engine oil warm up faster and achieve a better fuel economy by reduced mechanical losses. In the last decade supporting vehicle fuel economy improvement became the tertiary function of engine cooling system, but it came with a price: it has to be assessed running dynamic low to mid vehicle speed and engine loads, too far from severe driving conditions required to size radiators and fans.

A larger number of check and release sessions to grant both vehicle reliability and efficiency could impact new vehicle project timing and costs, but increased availability of affordable service facilities and gentlemen driving club circuits makes possible parallel testing by hot cell, proving ground and public roads runs, getting the best of each one.

In a single day you can cruise in the countryside and driving downtown in rush hours experiencing a real customer vehicle usage to calibrate engine cooling system software, then tow uphill or running power laps on a track to assess engine cooling system hardware. Hot cells offer tunable environment conditions and

mimic different road, vehicle and engine set up in a single location and are not restricted by weather and climate.

But in order to achieve the correct balance of test methods engine cooling performance simulations have to be improved too, extending their capabilities from system sizing and alternate components/subsystem checks to dynamic vehicle actions analysis – to prevent issues, provide guidance and allowing test result interpretation.

### What engine cooling simulation?

If passenger car engine cooling system simulations have to be more dynamic it means that they can be considered as a subcase of transient vehicle thermal management checks.

Therefore Flowmaster 1D CFD environment is appropriate to set up new capabilities and verify model building issues, being worldwide used by the automotive industry and proven effective for hydraulic and heat transfer studies. Compared to simulations for thermal management 1D CFD for engine cooling deals with higher flows and heat load, while vehicle speed profiles have to be estimated rather than imposed : compared with sizing simulations it's much more time consuming and labor intensive. In order to meet both reliability, compliance with project schedule and costs the “Keep It Simple, Stupid” or KISS approach is effective, so extended capabilities engine cooling 1D CFD models were built according the following guidelines:

- cooling airflow branch and each coolant branch sub-networks reduce all non-component related pressure losses to a single, overall discrete loss element;
- fan(s) and valve(s) driven by Flowmaster controllers, not by equations;
- simple lumped mass engine and thermal bridge loop to deliver heat rejections;
- Re-Nu heat map for air cooled heat exchangers extended for zero flows;
- vehicle speed, engine revs and heat rejection computed according vehicle and engine data outside Flowmaster environment, then included as time based inputs;
- exact engine cooling module layout to get component interactions.

### What for engine cooling simulation?

Considering functions of automotive engine cooling systems it seems reasonable investigating new simulation models for hot cell and track tests first. They are used to release engine cooling systems and set up targets and vehicle specifications by investigation of similar or similarly powered vehicles. Compared to real road testing both hot cells and tracks check a smaller range of driving conditions, but these ones are more challenging being unrestricted by traffic rules: additionally, they reduce safety issues, prevent security and press leaks about new models and allow quick repairs or tuning by their own facilities.



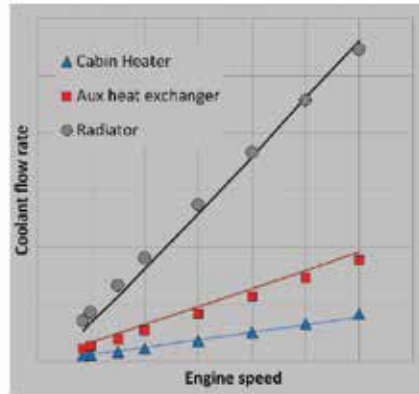
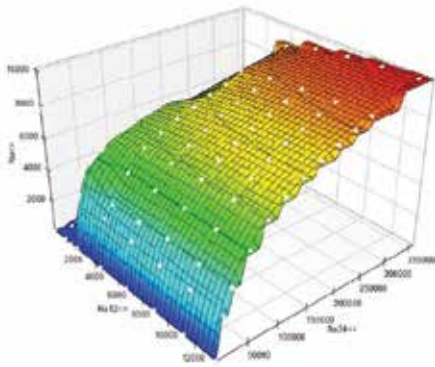


Fig. 4 - A collection of small, single purpose steady state simulation models assess reliability of main model features and heat maps reducing tuning loops and saving time

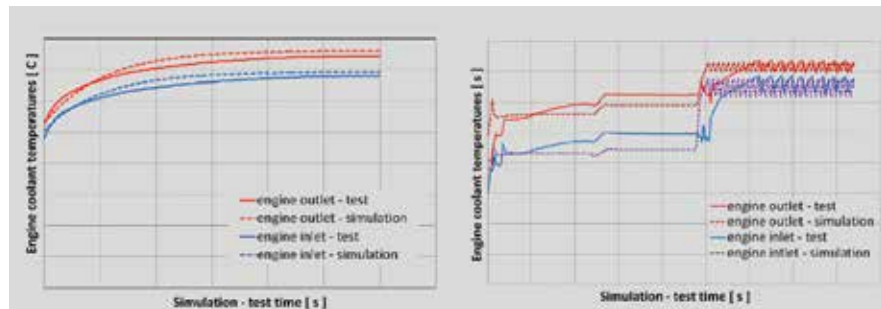


Fig. 5 - Release hot cell thermal tests results compared with simulation results

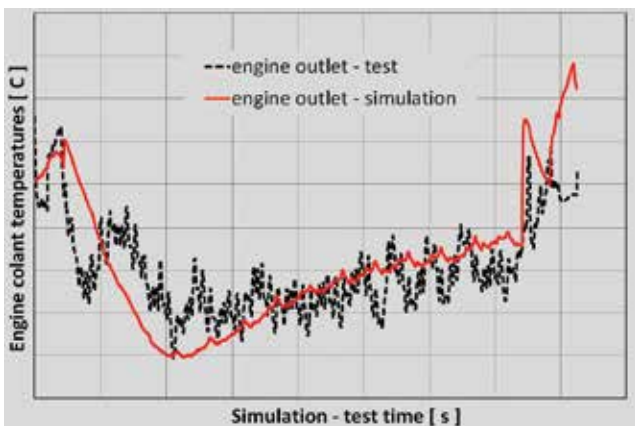


Fig. 6 - Predicted and recorded engine coolant temperature during a track test

Usually it's not difficult set up boundary conditions for 1D thermal models simulating hot cell testing but 1D thermal models simulating track runs need more efforts. If onboard recorded data are unavailable boundary condition should be set according expensive and time consuming full scale vehicle dynamics studies, but KISS approach and bibliography show a simplified yet effective approach under the following guidelines:

- vehicle is reduced to a slimy, lumped mass stick to the ground by tire grip only;
- vehicle does not accelerate or brake running bends;
- vehicle cornering speed is set according maximum tire grip and bend radius;

Track test event sequence as follows:

- idle leaving in the pits
- warm up installation lap
- power laps
- cool down lap
- hot soak, idle leaving in the pits

Once assessed flow reliability predictions every 1D CFD engine cooling model set up for transient studies should be thermally tuned and the simpler steady state hot cell tests results, if available, are well suited for this purpose. These tests produce a slow temperature evolution inside the engine cooling system, allowing refinement of fluid volumes and other non-fluid related thermal masses imposed inside components: additionally, steady state hot cell test results they can be used to double check engine nominal heat rejection map.

A test performed turning off and then turning on passenger cabin AC system, possibly including various vehicle speed, serves a similar purpose assessing reliability of predicted AC system heat rejection to cooling airflow upwind main radiator. This parameter is very important to achieve a good simulation precision for low vehicle speed cooling tests.

- vehicle accelerates and brakes running straights;
  - vehicle acceleration is limited by tire grip, even if engine traction would be higher;
  - vehicle brakes to match target cornering speed, using maximum tire grip.
- Block based scripts achieve a 4% to 12% maximum error estimating lap times by these simple rules, provided gear shift schedule, vehicle drag and engine power are available.

### Test case, simulation build up and tuning

An European market oriented, mid size car equipped with a high power output gasoline engine an a classical cooling system is used as a reference to develop 1D CFD models for hot cells and track test studies. Engine coolant components were set up considering nominal performances and heat rejection maps was nominal too, but extended to cover zero power and zero engine speed conditions to prevent solver numerical issues. Aside main simulation model a certain number of simpler auxiliary models were build and used to check out reliability of heat exchanger maps or provide "equivalent" pressure drops for coolant and cooling airflow branches. Then main model steady state, cold flow rate estimations results were compared with test bench and CFD flow results and despite a fair to good agreement was achieved appropriate corrections were implemented: as usually happens the largest deviations were found for low vehicle speed cooling airflow estimations.

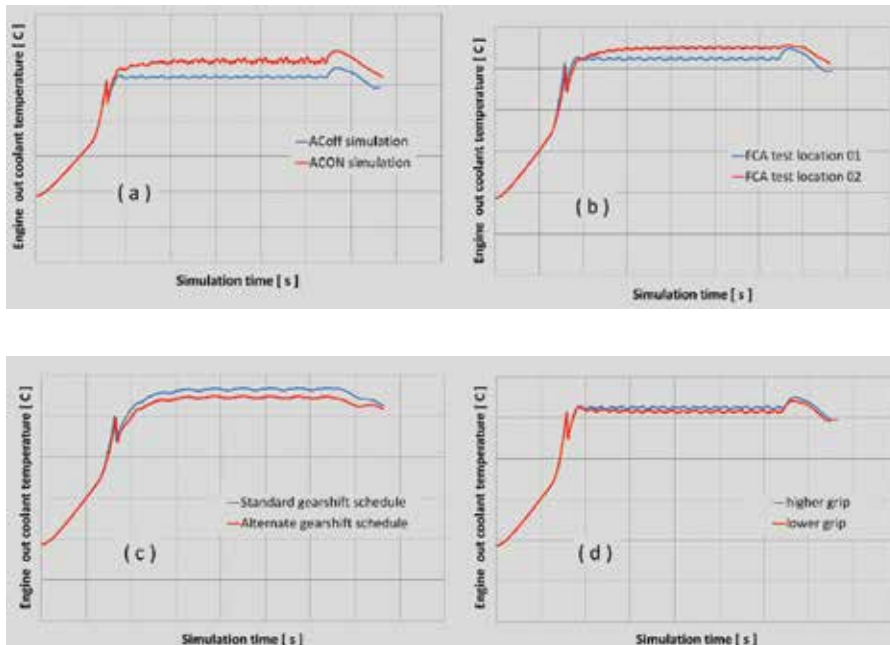


Fig. 7 - Some example of “what...if” comparison model runs: track simulations are performed modifying vehicle, transmission, tire or running a different track at all

### Predictive mode for new vehicles

1D CFD upgraded models can anticipate hot cell test results for a new vehicle because thermal network takes care of different engine cooling components, while effects of different specifications such as gear ratios improving fuel economy or increased towing capacity to meet customer expectations are accounted for as new boundary conditions. According preliminary evaluations track test results can be anticipated too, as long as modified engine, vehicle, track parameters are handled by block based vehicle performance simulations. Figure 7 shows comparison of coolant temperature prediction according: (a) tests performed with AC system engaged and disengaged; (b) tests performed running an European and a North American track; (c) alternate and standard gear upshift schedule; (d) nominal and reduce tire grip due to dust or rain on the track.

### Comparison of simulation and test results

After tuning 1D CFD engine cooling model was used to estimate temperatures running both hot cell and track tests oriented to final vehicle release. Predicted and recorded hot cell test results are shown in figure 5, being very close each other for the simpler test on the left. The more complex, three stage hot cell test on the right shows a fair enough prediction, the largest difference with experimental data does not exceed 6%. Simulation detects all events occurred, including thermostatic valve opening, transition between test stages and the fan cycling from full power to half power occurred during the final, most challenging stage.

Similar results are achieved running track simulations, as show in figure 6. This 15 minute long test is simulated using as boundary conditions reported vehicle speed and engine revs, calculated heat rejections and model calculated coolant and cooling airflows. Results are again fair enough, largest difference are found during event transitions and final hot soak.

### Summary. But not the end

For an increase in complexity, building time and labor 1D CFD Flowmaster full transient models improve evaluations of engine cooling system dynamic performances. However, usage so far suggests a certain number of improvements and developments:

- investigations of all classical engine cooling module layouts;
- investigations of all dual loop engine cooling systems variants;
- higher level control strategies for electrical coolant pumps and valves;
- better control strategies for active grill shutter devices;
- feasibility studies to include calculation of vehicle drag power and engine or transmission heat rejection inside Flowmaster models;
- elevation profiles for vehicle performance evaluation to take care of tracks such as Imola, Spa and Nordschleife and extend dynamic analysis to mountain road testing.



Fig. 8 - Introduction of grade effects extends engine cooling system dynamic simulations

N. Cauda, M. Colantoni, G. Gotta  
FCA Research and Development,  
Engine Systems

Ask the expert in 1D  
automotive simulation:  
Alberto Deponti, EnginSoft  
a.deponti@enginsoft.it

# Residual stress of railway wheels

## A predictive method

Starting from the requirements of the European standard EN13262 on the residual stresses of a railway wheel for its homologation, a virtual simulation of the heat treatment process is done in order to estimate these stresses. This simulation is done by using MAGMA 5 (Finite Volume), ANSYS (Finite Element) and the combination of both software. The residual stresses distribution is calculated according to the prescriptions of the AAR standard S-669 and is finally compared with the experimental results coming from the strain gauges measurement according to the European standard EN13262.

### Wheel and residual stresses

Railway wheel is one of the most critical component of a train from the safety point of view and the main issue is its structural design. So, the residual stress level due to production activities and mainly to the special heat treatment, consisting in an austenitizing followed by rim quenching and tempering, is to be considered an important parameters to be taken into account.

For this reason the applicable European standards prescribe to check the residual stresses on each new wheel design ("Strain gauge method" or "Ultrasonic method", that are both quantitative methodologies) and on each production batch ("Cutting method", that is only a qualitative method). Unfortunately, no input or suggestion is given by the above mentioned standards about the possibility of estimating the residual stresses before production starting, e.g. by process simulation. The study involves finding the best virtual approach, using two different software (ANSYS and MAGMA 5), to predict the final quality of the wheels.

### Process description

The wheel shall undergo a heat treatment that can be schematized as in the Figure 1. At the end of the heat treatment the residual stress level is similar to the one acting in the wheel when it



is in service. Between austenitization and tempering, a differential quenching ("rim quenching") is carried out. This quenching is obtained by putting the water into contact with the thread surface of the wheel only. In Figure at the top wheel section is shown in green color, while the area in contact with water is highlighted in red.

### Software simulation

#### Mesh and configuration

ANSYS analysis is performed on a 2D model of the wheel. MAGMA 5 analysis is performed on a 3D model of the wheel. Moreover in MAGMA 5 model is created also the volume of water and air needed to simulate the real configuration of the heat treatment.

The heat treatment parameters and coefficients are taken from the standard AAR S-669.



Fig. 1- wheel mean temperature during heat treatment



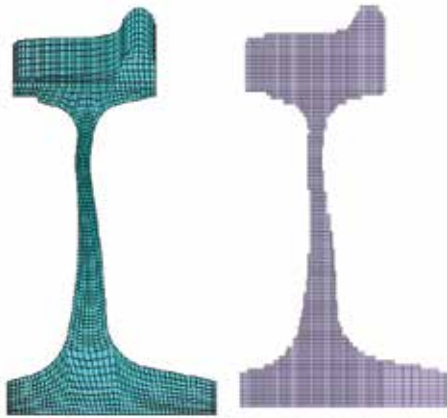


Fig. 2 - mesh of section of the wheel – ANSYS (left) and MAGMA 5 (right)

### Stress results

The residual stresses distribution obtained from the thermal and thermo-structural analyses performed with the two above mentioned software (ANSYS and MAGMA 5) is reported in Figure 3.

### MAGMA 5 + ANSYS simulation

ANSYS mesh is transformed into 3D by sweeping the 2D one. Using MAGMALink module, the results of the thermal analyses from MAGMA 5 are transferred into ANSYS. Then only the values related to the nodes of 2D mesh are taken into account and the residual stresses analysis is performed again in ANSYS.

### Results comparison

The results from all the simulations are compared with the values measured with strain gauge method on two test wheels (see Table 1).

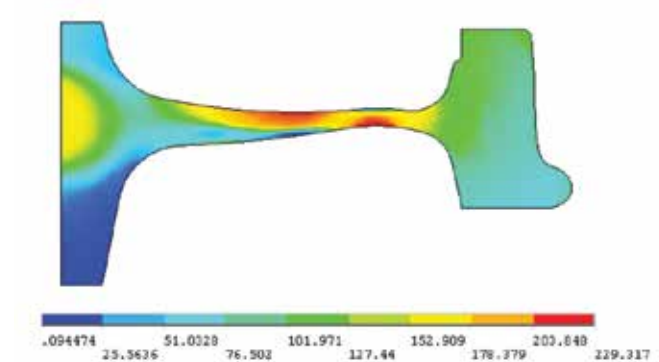


Fig. 4 - von Mises stresses in the wheel at the end of heat treatment  
MAGMA 5 + ANSYS

### Conclusions

The software programs ANSYS and MAGMA 5 appeared to be usable tools for the prediction of the residual stresses occurring in a railway wheel after heat treatment. Some large deviations between simulations and experimental results are recorded, but it can be noticed that the analyses carried out with the “mixed” MAGMA 5 + ANSYS approach are always either aligned with test results or conservative. Therefore, according to the obtained results the above mentioned “mixed” MAGMA 5 + ANSYS approach appears to be the best solution for this kind of analysis.

It must be considered that, as a first approach, the analysis were run with the default parameters provided by the software to consider the boundary condition and the effects of the water on the wheel surface.

A calibration activities of these coefficients, using an reverse engineering approach, is underway to refine the calculations and allow to estimate the described calculation to be 100% reliable.

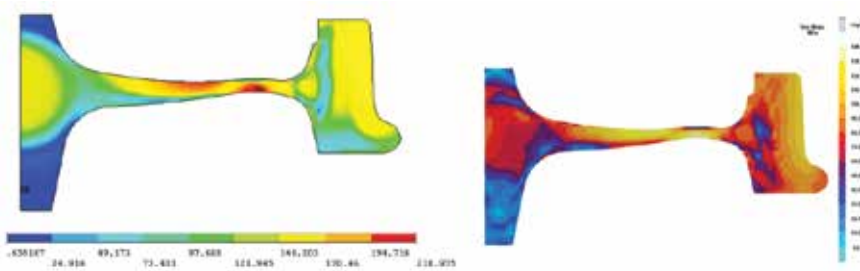


Fig. 3 - von Mises stresses in the wheel at the end of heat treatment, as obtained from ANSYS (left image) and MAGMA 5 (right image) simulations

Alberto Ronchi, Mirko Treachi  
Lucchini RS SpA

Ask the expert in the railway components stress simulations:  
Giampietro Scarpa, EnginSoft  
g.scarpa@enginsoft.it

Position	Radial coordinate	Stress	Residual stresses							
			Test 1	Test 2	ANSYS	Δ ANSYS	MAGMA	Δ MAGMA	MAGMA + ANSYS	Δ MAGMA + ANSYS
Outer web	225.1 mm	"Radial"	193.6	173.8	208.659	13.59%	126	-31.41%	224.8	22.37%
		Circumferential	96.7	84.1	82.341	-8.91%	56	-38.05%	88.5	-2.10%
Inner web	188.2 mm	"Radial"	31.2	37.1	107.932	216.05%	66	93.27%	100	192.83%
		Circumferential	31	33.5	23.963	-25.70%	19	-41.09%	54.1	67.75%

Table 1 - Results comparison (“radial” means principal stress along the local tangent direction to the wheel web in the cross-section plane) – “Magma” stays for “MAGMA 5”



# Development of a virtual calculation environment for multiphysics design & FEM analysis of large generators with particular focus on the electromagnetic analyses executed by means of proprietary algorithms embedded in the ANSYS Maxwell post-processing tool

In general, investigations using FEM software provide some benefits such as:

- accuracy and reliability of the calculation results;
- support from the software developers for solving complex and problematic FEM models and for getting custom macros ready.

On the other hand, the FEM investigation method can also lead to some drawbacks, such as:

- long calculation time;
- specific skills required for using the software, especially when a multiphysics approach is required to investigate different phenomena.

Aiming to reduce the impact of the above drawbacks, the following steps were undertaken in collaboration with EnginSoft Technical Support team:

- a custom multiphysics environment, based on Maxwell and ANSYS Workbench tools, including a simplified user interface was developed to permit even unskilled users to run simulations and determine some significant standard results by means of suited post-processing calculation
- some proprietary algorithms were embedded in this user interface to reduce the overall computation time.

The first part of this paper will describe how this smart user interface was tested on a test-case involving a 2D multiphysics analysis of a large synchronous generator currently manufactured by Ansaldo Energia. Then, in the final part of the paper, an “extension tool” for expanding the FEM multiphysics model from 2D to 3D will be presented along with some innovative 3D multiphysics electromechanical FEM analyses.



## Introduction

In fig. 1, is shown how the logic of the multiphysics calculation environment was organized behind the interface. The yellow blocks on the left side represent the CAD modelling tool, which can be carried on either by means of the UGS NX Plug-In available in ANSYS Workbench for connecting the CAD and the FEM environment, or directly by ANSYS Maxwell’s own CAD modeler. The green block in the center represents the

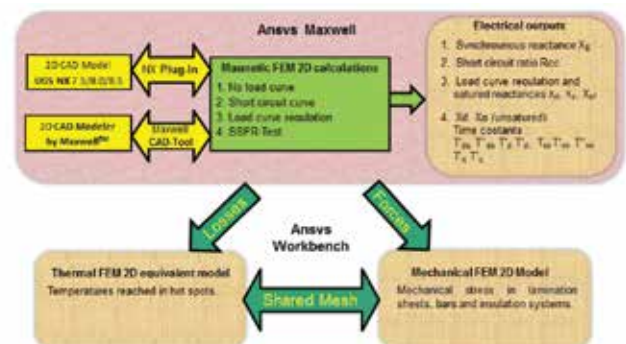


Fig.1 - General overview of the 2D multiphysics calculation environment

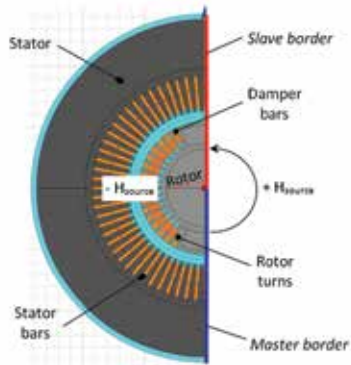


Fig. 2 – The 2D CAD/FEM model of a 180 degree section of the main region of the generator. Odd periodic boundary conditions have been applied to the Master and Slave borders

electromagnetic calculation environment implemented in ANSYS Maxwell which is the core of the entire multiphysics calculation platform, as it permits the calculation of all the characteristic curves of the generator; then by secondary post-processing analyses, performed by ANSYS Maxwell, the electrical parameters of interest for the generator's designer can be derived, obtaining:

1. the synchronous reactance  $X_s$ ;
2. the short circuit ratio  $R_{cc}$ ;
3. the load curve;
4. the unsaturated reactances and time constants calculated on both d-q axis.

After electromagnetic calculations have been run, the ohmic losses determined by the electromagnetic analyses can be transferred via ANSYS Workbench as input dataset for thermal analyses, which permits us to calculate temperatures reached in hot spots and sensitive positions. In a similar way, the output dataset related to electromagnetic and electrodynamic forces can be transferred via ANSYS Workbench as input dataset for FEM mechanical analyses, thus permitting us to calculate mechanical stress in lamination sheets, bars and insulation systems.

### 1. The CAD parametric modelling practise suitable for FEM model

The first necessary step to implement an efficient FEM model for design purposes is to establish an automatic virtual link between the FEM & CAD modelling environments. All the main geometric CAD parameters of the generator under investigation must be available in the FEM environment permitting us to evaluate the effects on the output dataset caused by editing CAD parameters. In addition, the availability of the CAD parameters within the FEM environment makes the implementation of optimization analyses, using optimizers embedded in ANSYS Workbench such as Optimetrics or modeFRONTIER, possible. In ANSYS Maxwell, two different approaches can be adopted for the implementation of the CAD/FEM parametric model of the generator (both conceptually reported in the yellow blocks in fig. 1):

1. using the UGS NX Plug-In, which permits the connection between the UGS NX CAD software and the ANSYS FEM software, UGS NX being the standard CAD modelling software adopted in Ansaldo Energia;
2. creating the design of the generator by means of Maxwell's own CAD tool by using some elementary forms such as circles, rectangles and polygons.

For 2D FEM modeling, the second choice of the two mentioned above has been adopted, eliminating the need to have further licenses, in addition

to the benefits of having all the CAD parameters directly available in the *variables\_page* of the project currently being carried out in ANSYS Maxwell. Such an approach has been also possible because the 2D CAD/FEM model has been suitably simplified, suppressing many CAD details such as chamfers or radii of curvature, as well as the involutes of the stator/rotor windings which do not contribute to the predominant phenomena occurring in the main region of the machine.

In fig. 2, the 2D FEM/CAD model of the generator obtained by means of a cross section orthogonal to the axis of the machine is reported. Only a 180 degrees section of the machine has been modelled as ANSYS Maxwell permits the application on the Master and Slave borders of odd periodic boundary conditions, thus reducing the model of the machine to only a half section of 180 degrees (as can be seen in fig. 2). The electromagnetic model takes into account only the active components of the generator such as conductive bars, damper windings, stator and rotor bodies, which are enough to calculate the main electrical parameters of the generator. Instead, for the 3D case, more complex geometries need to be modelled such as, windings involutes, radial ventilation ducts, phase connections, stator press plate and so on. Therefore for the 3D case, the NX Plug-In has been adopted to import complex CAD geometry to the FEM environment, which needs to be modelled by a skilled CAD designer working with real CAD software.

### 2. The proprietary algorithmic embedded in the FEM post-processing tool of ANSYS Maxwell

As mentioned in the abstract, investigations using FEM software provide some benefits such as the accuracy and reliability of the calculation results. However, the calculation time is a drawback which can lead most of the designers to prefer other methods of calculation, which are less accurate, but much simpler and faster to run. These include electronic spreadsheets containing a series of

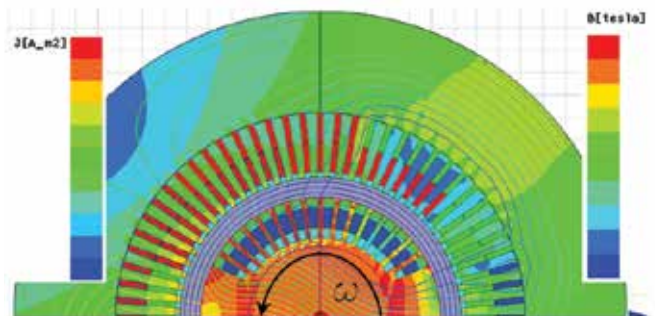


Fig. 3a – 2D transient FEM simulation: the electromagnetic fields distribution is recalculated at every rotor position analyzed during the rotor revolution

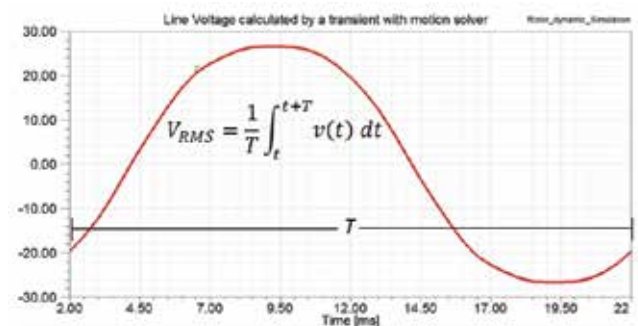


Fig. 3b – The Induced voltage is recalculated directly by FEM software at every rotor position analyzed during the rotor revolution



formulas validated by means of experimental measurements carried out during the years by the company. So since the FEM calculation platform here presented has to be used side by side with traditional methods of calculation such as the electronic spreadsheets, the FEM calculation time should be reduced as much as possible. To achieve this goal some proprietary post-processing algorithm has been embedded in the ANSYS Maxwell post-processing tool. In particular, a proprietary algorithm that analyzes the distribution of the vector potential in correspondence to the stator and rotor bars has permitted the calculation of the induced voltage without using the transient solver, but running only one magnetostatic calculation. This algorithm embedded in ANSYS Maxwell has been tested on a 2D FEM electromagnetic model of a synchronous generator manufactured by Ansaldo Energia, comparing the results of calculations with the real measured data obtained by means of the testing facilities provided by Ansaldo Energia. In addition, a comparison of the induced voltage calculated by means of the transient (fig. 3a-b) and magnetostatic analyses (fig. 4a-b) is here below reported as further evidence.

In both figures 3a and 4a the current density on bars and flux lines on ferromagnetic bodies are reported. While the first one reports the distribution of the induction field on the ferromagnetic bodies, the second one reports the distribution of the vector potential. This aims to explain the different ways to calculate the induced voltage, by means of the transient simulation and the magnetostatic approach respectively. Using a transient simulation, the induced voltage in each winding is provided directly by the FEM software which is able to compute the time trend of the flux linkage during the rotor revolution computing the induced voltage in each phase immediately (calculation results reported in fig. 3B for one phase).

The transient analyses are really helpful especially when transient phenomena such as short circuits, load rejection and asymmetric supply conditions need to be analyzed. When the generator works at steady state, it could be helpful to keep in mind that:

- the machine stator and rotor core has symmetrical structure;
- the 3 armature windings have symmetrical structure;
- effect of eddy current and hysteresis are negligible.

This characteristic permits us to develop some algorithms able to derive time dependent results by running only one magnetostatic simulation. One of these algorithms has been developed ad hoc to calculate the induced voltage on stator phases; it executes the calculation of the direct  $\varphi_d$  and quadrature  $\varphi_q$  component of the resulting total flux  $\varphi_r$  (conceptually shown in fig. 4a) in relation to the armature windings layout. This specific calculation practice and more generally other customized formulations have been implemented in ANSYS Maxwell by means of the "Fields Calculator" post-processing tool (reported in fig. 4b) which permits us to write wide facilities of post-processing formula of the electromagnetic fields calculated by the FEM simulation.

Once the calculation of the direct  $\varphi_d$  and quadrature  $\varphi_q$  component has been done, the calculation of the RMS value of the induced voltage in each phase can be computed by the following formulas:

$$v_d = f_1(\omega, \varphi_q), \quad v_q = f_2(\omega, \varphi_d), \quad V_{RMS} = \sqrt{v_d^2 + v_q^2}$$

where the functions  $f_1$  and  $f_2$  are proprietary formulation developed ad hoc. To validate this calculation practice the RMS value of the induced voltage obtained by means of the transient and magnetostatic simulations, for a wide range of operative conditions (no load, rated load, many leading and lagging load cases) has been compared and a difference in the range of a few per cent (roughly 2÷3%) has emerged. Also further comparison with measured data obtained by Ansaldo Energia testing facilities has been implemented and a good match has verified.

### 3. The interface developed for permitting even unskilled users to run FEM Electromagnetic calculation

As mentioned in the abstract, an annoying problem encountered in the use of FEM methods for design purposes is that the interface of the FEM software obviously can not be customized to the needs of each end-user. In our case, in order to satisfy this need, a graphic interface which hides the FEM model has been created, and that permits the

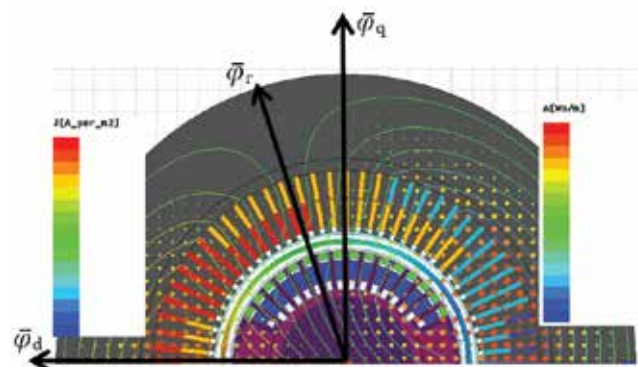


Fig. 4a – 2D Magnetostatic FEM simulation: electromagnetic fields distribution calculated @ one rotor angle position

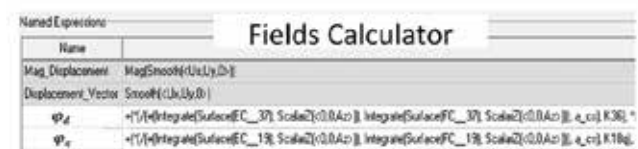


Fig. 4b – Fields Calculator post-processing tool of ANSYS Maxwell: the formulas embedded for calculating  $\varphi_d$  and  $\varphi_q$

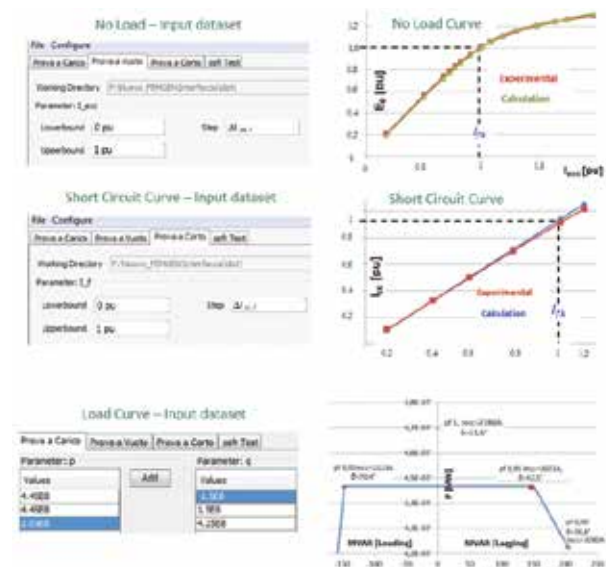


Fig. 5 – The customized end-user interface developed to permit even unskilled user to run FEM calculations

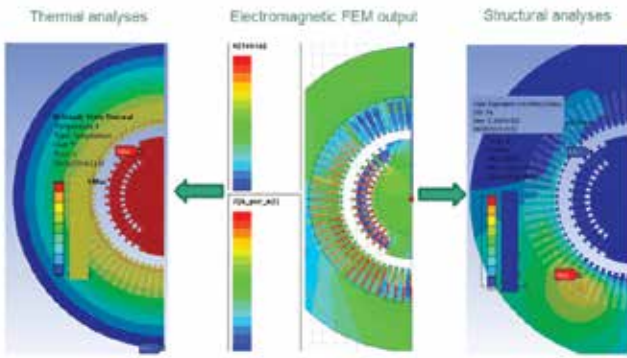


Fig. 6 – The 2D CAD/FEM multiphysics model of a 180 degree section of the main region of the generator to run thermal and structural analysis starting from the electromagnetic simulation

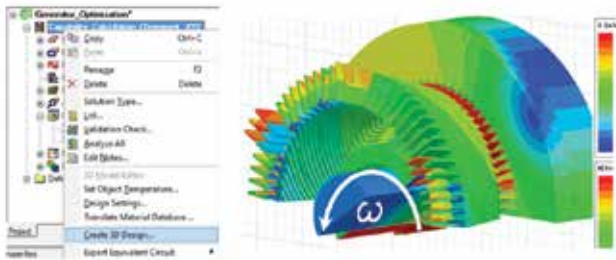


Fig. 7 – The distribution of the magnetic induction on ferromagnetic bodies and the current density distribution on windings calculated by the 3D FEM electromagnetic model

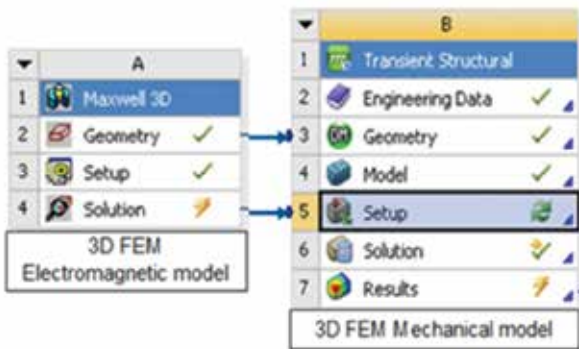


Fig. 8 – The ANSYS Workbench interface for transferring the electromagnetic forces as input dataset for structural analyses

end-user to run FEM calculation just by editing the input dataset on the suitable boxes. In the fig. 5, an example of how to work with the FEM software for calculating the *No Load Curve*, the *Short Circuit Curve* and the *Load curve* is reported; it should be really clear how this interface is user-friendly, even for an unskilled user on FEM software. The *No Load* and *Short Circuit Curves* can be used for the calculation of the synchronous reactance  $X_s$  and the short circuit ratio  $R_{sc}$ , while the load curve provides the excitation current and the load angle for every supply condition required by the customer, thus permitting us to establish the stability of the alternator. All the output dataset reported in the graph of fig. 5 has been compared with measured data obtained by the testing facilities of Ansaldo Energia. To calculate the load curve, it has been necessary to embed within the FEM software an iterative procedure followed by an optimization analysis. This automatic calculation practice has been programmed by means of an Open Source program language compatible with ANSYS Maxwell, while the optimization analyses can be implemented just using the Optimetrics, the standard optimizer integrate within the FEM software.

This complex calculation practise has been necessary because the right set of excitation currents and load angles which will permit to the generator to supply each set of power output required (the P, Q values @ fixed voltage) is the unknown of the problem, which could be calculated only if the reactances ( $X_d, X_q, X_{d'}$ ) of the machine were known at the operative condition under test. Since the reactances depend, in a non-linear fashion, on the power output supplied, a possible solution to the problem in these cases is the use of iterative practice and optimization analyses as it has been implemented to calculate the load curve .

#### 4. The 2D FEM multiphysics calculation environment

As can be seen from the flowchart reported in fig.1, using the ANSYS Workbench interface a real multiphysics calculation environment has been implemented. Mechanical, thermal and electro-magnetic FEM models share the same CAD model; the output dataset of the magnetic calculation as forces and ohmic losses are considered as input dataset respectively for structural and thermal analyses. Following such an approach, most of the design parameters can be calculated by running just one multiphysics FEM calculation. In the fig. 6 the results of a 2D test-case FEM multiphysics calculation are reported, aiming to show how the FEM multiphysics model is organized behind the interface. The electromagnetic 2D FEM model (in the middle of the fig. 6) has always been run first at the operative state of interest, obtaining as a result the electromagnetic fields such as induction, current density, eddy currents, electromagnetic forces and so on; then sharing the same CAD model between thermal and structural FEM environments, the “loads” such as forces and losses, have been used as input dataset for structural and thermal analyses respectively.

Then reasonable constraints and boundary conditions are added to the structural and thermal FEM model thus permitting us to run a multiphysics simulation as near as possible to the real state of the machine, optimizing the calculation time. In fact, for example, when the velocity and the pressure of the refrigerant fluid are well known, by means of a detailed analyses of the geometry on where the refrigerant will flow, the thermal analyses can be simplified by applying to the border of each body its own coefficient of convective thermal exchange; this practice has been validated with the support of the temperature measured data obtained from the diagnostic system which was installed inside the generators from so many years ago. In the left side of fig. 6 the results of a 2D FEM steady state thermal analysis that follows this kind of approach can be seen. In a similar way, for the structural analyses, some considerations, about how the single bodies can move each others, lead us to introduce in the FEM mechanical model some fixed constraints, as for example on the external circumference of the stator core, which is effectively fixed to the frame. Then, depending on the aim of the structural analysis, either the electrodynamic forces on the stator bars or the electromagnetic forces on stator and rotor teeth can be imported as input dataset for the structural analysis. In the structural analysis reported on the right side of fig. 6, the distribution of the mechanical stress, calculated on the insulation systems of the stator bars using the electrodynamic forces on them in a particular supply condition under test, can be seen. This distribution of the mechanical solicitations, reported in terms of Von Mises stress, can be helpful for the designer, especially for sizing the components of the generator.



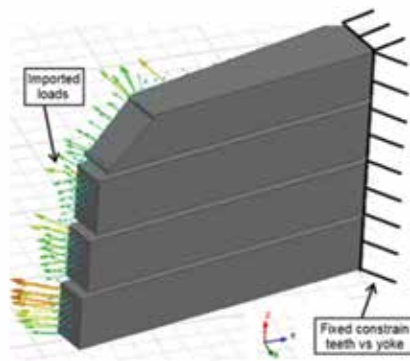


Fig. 9 – The 3D FEM Multiphysics modelling for structural analysis of the laminations at the end-region of the generator

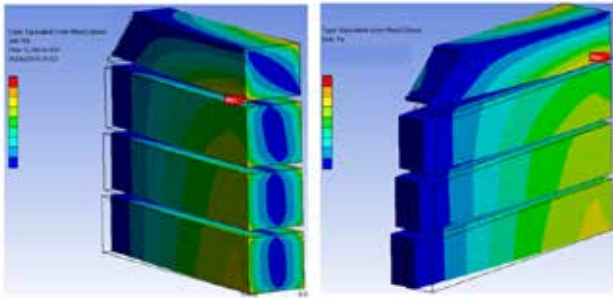


Fig. 10 – The Von Mises stress and deformation calculated on the laminations at the end-region of the generator

## 5. The “extension tool” for expanding the FEM multiphysic analyses from 2D to 3D model aiming to analyze dangerous phenomena at the end-regions

In this chapter the approach taken to extend the FEM analysis from 2D to 3D model will be presented. The 3D analysis is fundamental as the flux leakages due to the winding involutes create undesired and extremely damaging effects such as axial forces and additional eddy currents, in particular on the laminations located at the end-region of the stator core. These 3D analyses have been run at roto-dynamic state, modelling the rotor components as objects in rotation @ 3000 rpm, then synchronizing the rotor revolution with the rotating magnetic field created by the alternative currents that flow through the stator bars. The load angle, the excitation and armature current have been assigned in a way to reproduce the under-excitation supply state, where the dangerous phenomena such as eddy currents and axial forces are most intensive. The 3D model can be created starting from the 2D model, by extruding the 2D cross section of the generator along the axial direction, and by importing the 3D CAD model of the winding involutes from the 3D CAD modelling environment. The involutes, because of their complicated shape, need to be modelled by a skilled CAD modelling designer.

In fig. 7, the distribution of the magnetic induction on ferromagnetic bodies, and the current density distribution on windings calculated by the 3D FEM electromagnetic simulation are reported for one of the rotor positions analyzed. Using the same approach described in the previous chapter for implementing the 2D FEM multiphysics model, the “Loads” in terms of forces calculated by the 3D FEM roto-dynamic electromagnetic analyses can also be considered as the input dataset for structural analyses using the ANSYS Workbench interface (see fig. 8).

The results obtained by the 3D FEM electromagnetic calculation determine different conditions for the stator teeth, as some of them turn

out to be extremely subject to larger forces. Therefore, attention has been focused only on these stator teeth, i.e. the ones that mark the boundaries between the sector where bars belonging to different phases are deployed in the bottom layer. Then, due to the physical symmetry of the machine, and to the supply conditions considered, the field map distribution at any location in the stator should turn out alternative, i.e. at any time instant the value of each field component should be the opposite of what it was half a period before. This implies that the forces acting on ferromagnetic stator parts, such as teeth, should actually turn out to be periodic with a period equal to half of the electrical period. Therefore, according to the previous assumptions about the symmetry of the magnetic field distribution, with the aim of reducing the time calculation, the 3D FEM mechanical model considers only the stator tooth most exposed to the magnetic forces while the other ones can be suppressed from the model, as has been done in the model reported in fig. 9. The vector plot reported in this figure shows the distribution of electromagnetic forces which are acting on laminations at one of the time instants analyzed. To calculate the maximum mechanical stress encountered in the laminations during the rotor revolution, a transient mechanical analysis has been run by using the time trend of the 3D electromagnetic forces distribution calculated by the 3D FEM roto-dynamic electromagnetic simulation as input dataset. To suppress the stator yoke from the 3D FEM mechanical model, a bond of fixed constraint at the base of the tooth has been assigned as shown in fig. 9, thus optimizing the calculation time.

This practise has permitted a large reduction in the calculation time without losing any fundamental details for the simulation. In the fig. 10 the results of the 3D electro-mechanical FEM analysis can be seen. To report the mechanical stress status on the laminations, the Von Mises

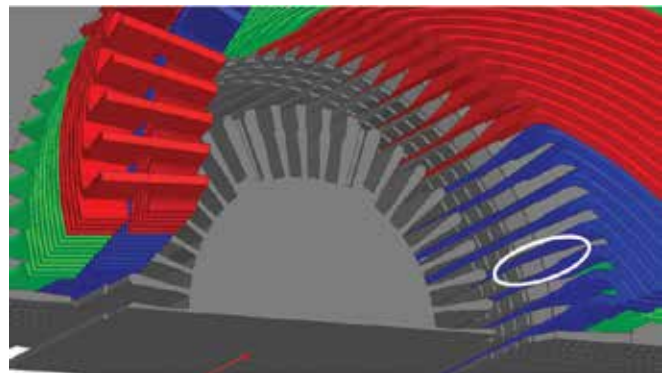


Fig. 11 – The end-region of the generator with focus on the laminations where the eddy currents will reach the max magnitude

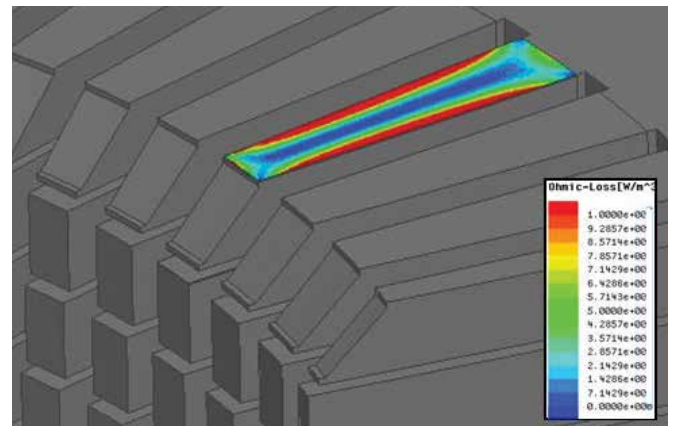


Fig. 12 – The ohmic losses distribution on the split thin layer, calculated by means of the 3D FEM roto-dynamic model on the hottest laminated sheet



Stress tensor has been calculated for every time instant investigated and the worst case is reported in fig. 10. The red label points to the area where the maximum mechanical stress solicitation can be detected during the rotor revolution. (In the two pictures of fig. 10 the deformation of the stator teeth has been greatly amplified).

Another important phenomenon, which is fundamental to keep under control for the safety of the generator, is the generation of the hot spots created by the eddy currents distribution on the outmost laminations stacked at the end region, which are the most exposed to the axial flux density distribution. The equations that govern the generation of the Eddy-Current are here reported:

$$\begin{cases} \nabla \times \bar{E}(x, y, z, t) = - \frac{\partial \bar{B}(x, y, z, t)}{\partial t} \\ \nabla \times \bar{H}(x, y, z, t) = \bar{J}(x, y, z, t) \end{cases}$$

They can be complex to solve even in simplified cases; so for the complicated geometry of the end-region of the generator (shown in fig. 11), considering also the non-linear fashion of the BH curve of the laminations, the 3D FEM analyses is necessary to calculate the eddy currents distribution on the laminations at the stator end-region.

To investigate this phenomenon, a 3D FEM electromagnetic & thermal calculation practice has been developed ad hoc by modelling some laminations as they really are, i.e. splitting a thin layer from the non-laminated body, which are normally used to model the laminations structure by means of 3D FEM and CAD software. So, a 3D FEM electromagnetic roto-dynamic simulation has been run to compute the eddy currents distribution and thus the specific losses dissipated within the split thin layer. Because different conditions for each stator tooth have been detected by the 3D FEM electromagnetic calculation, the

calculation of the eddy currents that flow through the laminations can be executed only on the stator tooth most exposed to the axial flux density, considering it as the worst possible case.

The results of the 3D FEM ohmic losses calculation obtained by means of this 3D FEM modelling practice is reported in the picture in fig. 12. In this picture the magnitude of the ohmic losses caused by the eddy currents flowing through the thin ferromagnetic layer at one of the time steps analyzed is reported and it has been considered as the heat source for the thermal analyses, keeping in mind the well-known Joule formula as follows:

$$P = \rho \cdot \frac{1}{T} \int_t^{t+T} J^2(t) dt \quad \left[ \frac{W}{kg} \right]$$

By the volume integration of the Joule formula on each mesh element, the heat sources generated in each finite element will be provided. Once the heat sources have been calculated and transferred as input dataset for the thermal analyses via ANSYS Workbench interface (as shown in fig. 13), the thermal convective exchange coefficients have been applied on each surface of the lamination sheet, keeping in mind the flow rate, the velocity and pressure of the refrigerant fluid in correspondence of the lamination considered.

Subsequently, the 3D FEM thermal calculation was run and the temperature reached at the steady state in the hot spot can be seen in figure 14b. The designer of the generator may use this electromagnetic and thermal coupled analysis to optimize the design of the laminations at the end-region to hinder the flow of the eddy current within them.

### Conclusion

A customized end-user interface has been developed ad hoc in order to permit even unskilled users to run the 2D FEM electromagnetic calculations necessary to calculate the electrical parameters of interest for the design of the generator (the synchronous reactance  $X_s$ , the short circuit ratio  $R_{cc}$ , the load angle & the excitation current to satisfy the load request).

A proprietary algorithm to calculate the induced voltage at steady state, by running only one magnetostatic simulation has been embedded in the ANSYS Maxwell post-processing tool, thus avoiding a transient FEM analysis, which running generally requires more calculation time.

By means of the ANSYS Workbench interface, for both the 2D model and the 3D model, some multiphysics simulations have been implemented in order to calculate the thermal and structural output dataset, such as the temperature in hot spots and the mechanical stresses on laminations, stator bars and the insulation system, which are fundamental to complete the design of all the components of the generators.

Iterative procedure and optimization analyses have been implemented to calculate the load curve by means respectively of open source program languages and Optimetrics, the optimizer integrated within ANSYS Maxwell.

*Ing. Fabrizio D'Agostino, Ing. Roberto Biondi,  
A.S.EN. Ansaldo Sviluppo Energia - Ansaldo Energia Group*

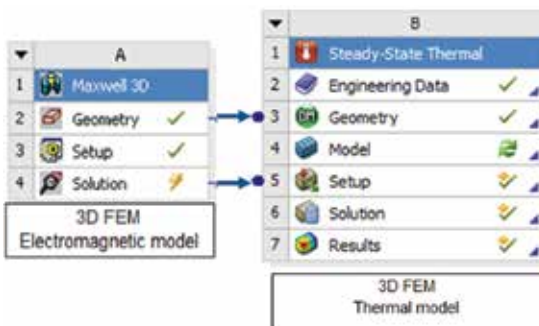


Fig. 13 – The ANSYS Workbench interface for sharing the 3D CAD model and transferring the thermal loads for thermal analyses

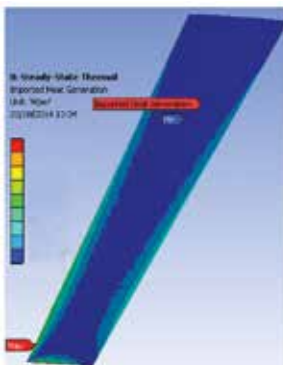


Fig. 14a – The heat sources obtained by the 3D FEM electro-magnetic analyses

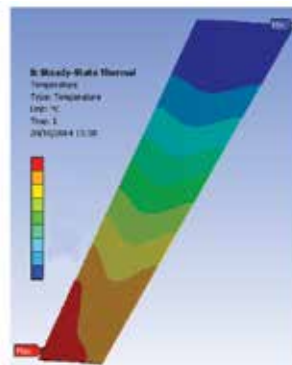


Fig. 14b – Temperature calculated on the lamination by the 3D FEM multiphysics analyses



# Flowmaster used for thermo-hydraulic flow simulations in STX France ship networks

## A tool for network design

During the design phase of the ship, the engineering software Flowmaster is used to simulate many thermo-hydraulic flow networks.

It enables the engineer to compute all the thermo-hydraulic properties of a steady state system and represent the nominal behavior of the system. It is also possible to follow all the thermo-hydraulic quantities during transient scenarios. The transient capability represents more specific operations: water hammer, valve opening/closing, pump start / trip, priming, etc. Finally, Flowmaster is also used to simulate compressible and incompressible flows for single-phase and two-phase fluids (water, oil, fuel, gas, air, etc.).

The aim of this kind of modeling is, at first, to check the sizing of pumps, heat exchangers, etc. Then to validate the pressures, flow rates, velocities, temperatures of the fluid at any point in the network and to ensure the system operates properly. The final goal is to optimize pipe diameters and the equipment's power to improve the overall energy balance of the ship.



## Examples of networks for steady or unsteady state incompressible flows

### Potable water system:

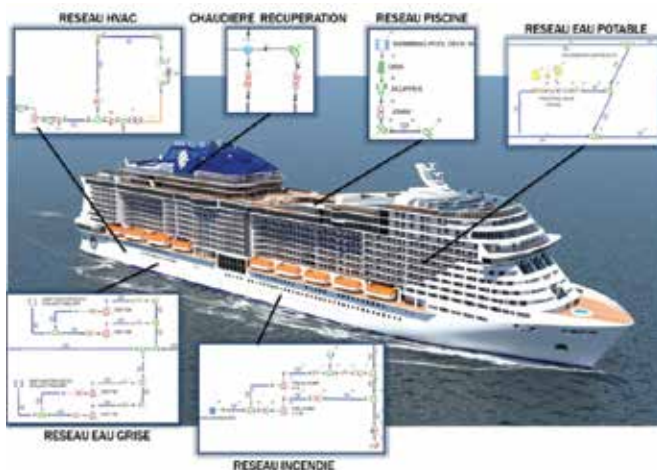
- Validation of available pressures in the distribution loops pipes and analysis of the hydraulic behavior during the variations related to the peak flow.
- Balancing of recirculation lines to maintain hot water temperature and to ensure the chlorine renewal in the water at any point of the ship.
- Checking the range of the pressure regulators and safety valves.

### With heat exchanger solver:

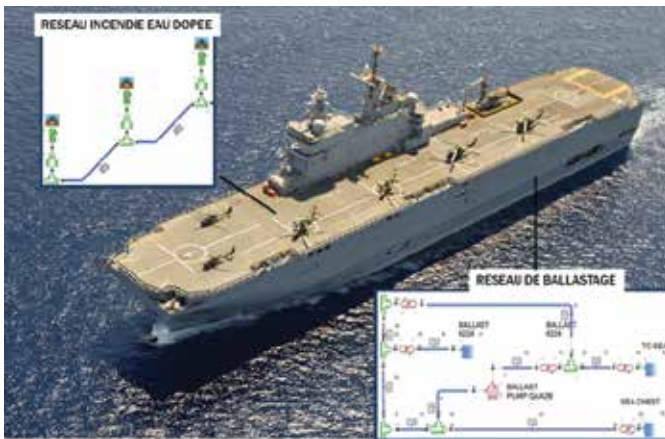
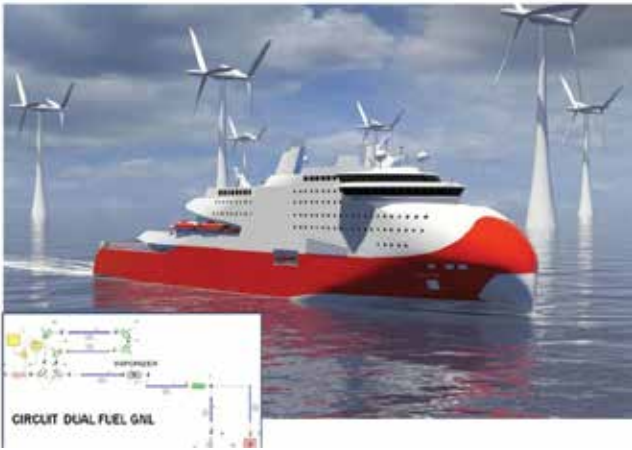
- Thermo-dynamic study of the sanitary technical cabinet in order to analyze the behavior during simultaneous utilization. (pressure/temperature fluctuation).
- HVAC system – cold water/hot water:
- Adjustment of control valves for the air conditioning fan coil unit. (Flow balancing simulations).
- Validation of total manometric head of pumps and of pressures available in any point of the ship.

### Ballasting systems

- PID control of flow rate regulation valve for ballasting and deballasting phase.
- Hydraulic behavior analysis and time estimation during docking and undocking operations.
- Validation of Net Positive Suction Head for Ballasting pumps.







#### Fire Fighting systems

- Verification of flow rates/pressures in the ship's fire hydrants according to regulation.
- Validation of homogeneous repartition of doped water in flight deck's.
- Simulation of the transient behavior of the network and checking absence of water hammer during the valve closing (Priming simulations).

#### Swimming pool systems

- Hydraulic behavior analysis during fast emptying of swimming pools, observation of water hammer and flow regime variations phenomena.

#### Refrigerating system of propulsion engines/diesel engines

- Balancing of the system according to flow rates imposed by the heat exchangers.
- Validation of total manometric head of pumps.

#### Sewage treatment

- Checking the different operating points of pumps according to discharge points (treatment, bunker, sea).
- Behavior analysis during simultaneous startups of discharge pumps towards sewage treatment unit.

#### Energy recovery systems of diesel engines

- System balancing according to flow rates imposed by the heat exchangers.
- Analysis of system's thermo-hydraulic behavior according to load variations of diesel engines (tropical configuration, winter, dock, cruise speed, etc.)

#### Flue gas cleaning systems

- Analysis of scrubbers' water rejections.

### Examples of networks for steady or unsteady state compressible flows

#### Fire Fighting system by CO<sub>2</sub>

- Simulation of CO<sub>2</sub> bottle opening; temperature variation analysis on the inner surface of pipes.

#### Hull lubrication system by air injection

- System balancing to ensure necessary flow rate in furthest pipe.

#### LNG System

- Behavior analysis of valve exhausts on LNG tank
- Gas leak simulation on supply pipes.

### With two-phase module

#### LNG System – Supply of gas engines

- Analysis of liquid phase transition and gas phase to supply “dual fuel” diesel engines with gas.
- Simulation of diesel load variations.

#### Steam system

- Simulation of superheated steam system towards turbine.
- Thermo-dynamic analysis of the thermo-siphon on the energy recovery boilers.

### Helping process improvement

During a testing phase, while the ship is in the dock, Flowmaster is used to perform predefined simulations to adjust the control valves and P.I.D. parameters. It is also used to check the simulation scenarios and to verify the predicted results with test data.

These models are essential in simulating extreme damage load cases and provide design improvements (ex: LNG study, Combined steam cycle).

### A tool to fostering innovation

The International Maritime Organization has promulgated new regulations which will prohibit the diffusion of sulphur in European seas by 2015.

These new requirements will force ship owners to use fuels which produce exhaust emissions with a very low sulphur content.

Ship owners have long been looking for a way to reduce their fuel bills and to optimize their ships' energy performance.

STX France has been trying to develop its knowledge in the eco-conception domain in order to design a new generation of passenger ships to meet the future environmental requirements, while working on operating cost reduction and on improving performance.

The aim is to provide ships which are both more ecological and economical.

Flowmaster and its two-phase module is perfectly adapted to find these system performance improvements.

New studies related to the reduction of ships' ecological footprint are currently being carried out. These are devoted to the modeling of a hybrid system fuel/LNG plant for natural gas supply of “dual fuel” engines, the modeling of the energy recovery system for diesel engines and the modeling of flue gas scrubbers.

François MARTINEAU - STX France  
francois.martineau@stxeurope.com

Ask the expert in Flowmaster:  
Romain Soulier, EnginSoft France  
r.soulier@enginsoft.com





# Piping afloat: Using CAE to calculate piping systems onboard ships

One major design aspect of a ship is the complex piping system onboard. These piping systems meet requirements during ship operation by handling ballast-water, fuel, ventilation, various cooler loops, etc... , but also for meeting needs when handling liquid cargo. The design objective for a ship includes minimizing weight and volumetric impact, but also to optimize the piping system's performance. Figure 1 shows the extent of piping that literally outlines the shape of the ship.

Salt Ship Design AS located in Leirvik, Norway, designs offshore and specialized vessels by combining well-proven solutions with innovative design by applying the best available technologies to the problem. Salt Ship uses state-of-the-art 3D tools for both design and visualization. Advanced CAE tools such as Flowmaster are used to optimize the onboard piping system. This article highlights the typical analyses performed using Flowmaster and 3D CFD for engine room ventilation.

## 3D Pipe layout imports

Given that space is precious, the engineer establishes the 3D piping layout early in the design phases. Flowmaster allows the import of 3D piping lay-outs, reducing model development time considerably. The added benefit being that the model includes all components, with their associated losses, from the very onset of when deciding pipe diameters and pump specification.

Figure 2 displays details of the same piping systems as seen in the Flowmaster GUI and in the 3D geometrical model. The numerical



analysis gives the maximum pressure in the shown system for so-called overflow conditions, a condition characterized by liquid, e.g. during bunkering, filling the air vent system. An unlikely scenario, but required in order to meet class requirements.

## 3D evaluations of flow

The air vent system supplying the engine rooms provides fresh and cool air as a way to remove heat from hot surfaces, but also to

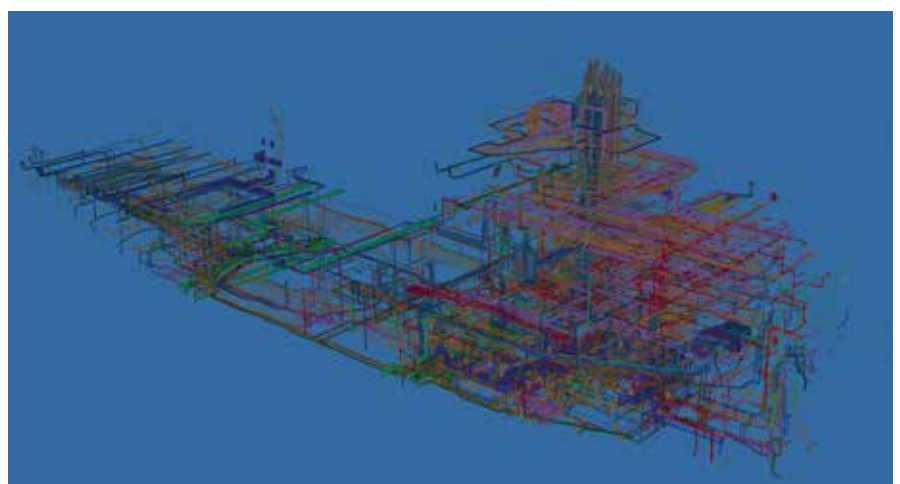


Fig. 1 - Example of Salt Ship Design solution. The extent of the piping exposed when removing the hull and internal walls

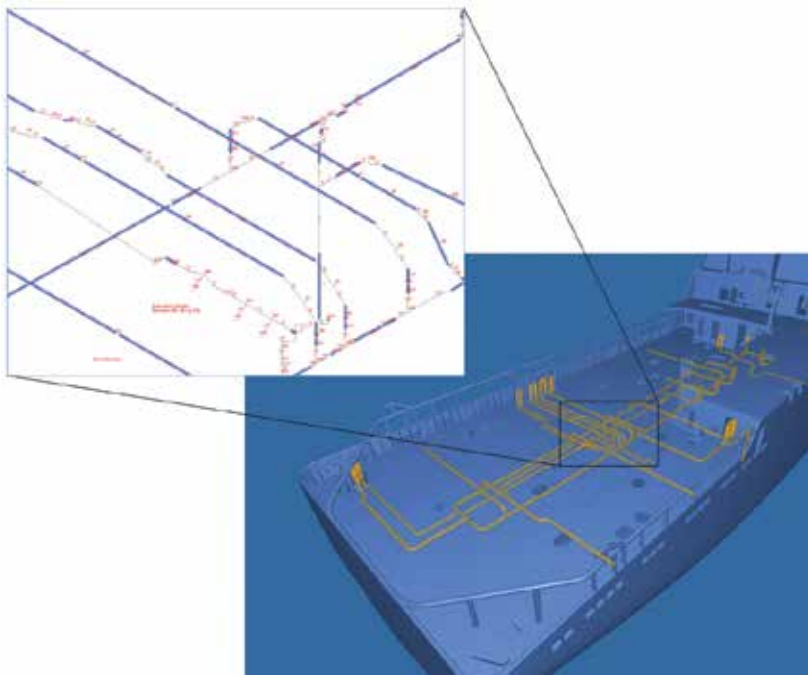


Fig. 2 - Piping sub-system aboard ship exported to Flowmaster for further analysis

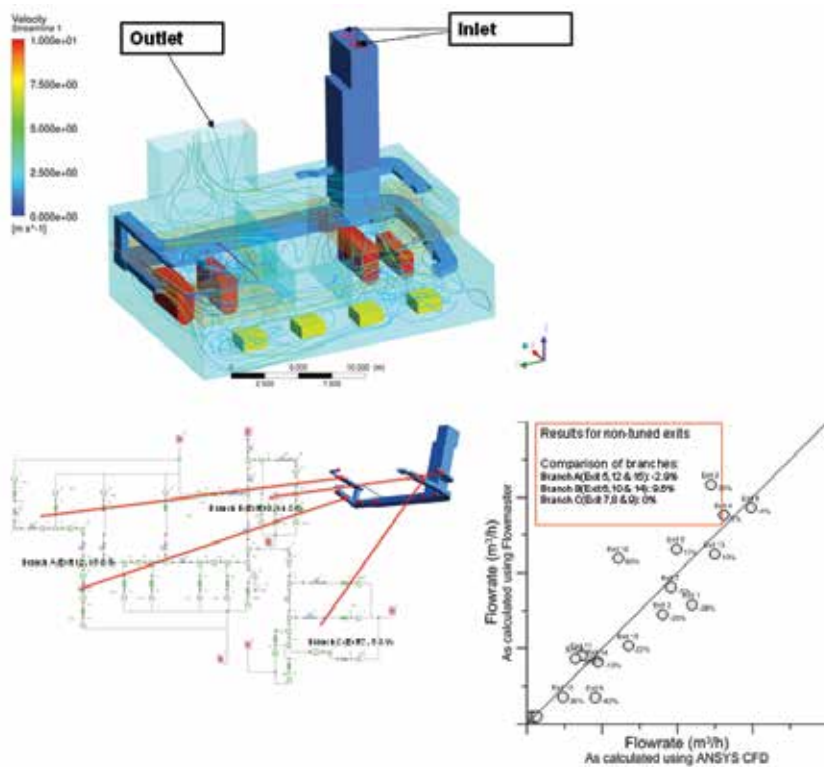


Fig. 3 - Calculation of engine room venting system, the corresponding Flowmaster model and the comparison between the two approaches

provide direct supply to air intakes at engines. The huge dimensions of the main ducts often makes the normalized distance,  $L/D$ , between components, such as various branches, less than unity. One consequence being that 3D flow effects dominate much of the overall performance, i.e. effects not necessarily captured well by the 1D approach offered by Flowmaster.

For the evaluation of the engine-room ventilation system, Salt Ship initiated a 3D study, with one important aim: to evaluate their own design methodology while also evaluating the analysis possible using Flowmaster. The 3D model contained a representation of the engine room including heat sources and flow sinks, while the post processing focused on the flow through each exit of the duct system as well as providing qualitative explanations for deviations in expected flow rates at exits.

A comparison between the solutions obtained by the 3D CFD and Flowmaster model showed discrepancies. By investigating the 3D model, the likely explanation of the discrepancy being strong local 3D effects at the branches of the duct system. From a global perspective, the flow distribution between branches compared well for the two approaches. One benefit with Flowmaster relates to the flow-balancing capabilities included. Having adjustable air vents, or outlets, will allow Flowmaster to provide guiding for adjustment.

Figure 3 shows the qualitative results from the 3D modeling, and the comparison between the two approaches.

### Summary

For one design delivery more than 30 different piping systems, require special attention for minimizing weight and volumetric impact, but also for the correct selection of critical components such as pumps. All for reducing the risk of not meeting expected performance after installing equipment at the yard. For the work flow at Salt Design, the iterative change of pipe-layout often require a rapid performance evaluation. As of now, the set-up at Salt Design allows an effective transfer of piping topology and dimensions to tools such as Flowmaster. The use of 3D CFD adds value by providing information that add to best-design-practices for flow systems where 3D effects are present; such as the air vent systems for engine rooms.

Morten Kjeldsen – Flow Design Bureau AS  
 Christoffer Järpner – EnginSoft Nordic

Ask the expert in piping 1D simulation:  
 Alberto Deponti, EnginSoft  
[a.deponti@enginsoft.it](mailto:a.deponti@enginsoft.it)





## Design of the Roller Coaster LOOP 395

The design of a Roller Coaster is always a new challenge. Such a design is a “mixture of many ingredients” and has to meet both structural-dynamic and safety requirements. Not only that, the Roller Coaster should be an original experience that provides a thrill to the rider.

So, we can say Roller Coaster designers have to be smart people, since they need both strong technical skills and creativity to create such an attraction.

These goals require long feasibility analyses and depend on the good sense of the designer. EnginSoft has been collaborating with Ride Tek Engineering (RTE) for many years now, developing an integrated CAD-FEM procedure to allow the virtual prototyping of any Roller Coaster. Further details are presented in the article below, with particular reference to Loop 395 Coaster, commissioned to Ride Tek Engineering by IE-Park. IE-Park are manufacturers of a wide range of products suitable for amusement and theme parks, family entertainment centers, shopping malls and many other leisure facilities.

Loop 395 Coaster is a traditional sit-down coaster characterized by a loop. The length of the coaster is about 395m, which is where the name “395” comes from. The ride is about 18 m high. Plan layout can be contained in an equilateral triangle of 75-80m side. The ride is entirely made of carbon steel S355 and its structural members are made from circular hollow tubes and plates. Loop 395 is another classic example of IE-Park/RTE interaction with EnginSoft. Typically, EnginSoft enters into the design phase after a feed phase of ride geometrical/emotional layout has been defined by both RTE and IE-Park.

EnginSoft contributed in the past to the



development of a specific CAD-FEM procedure that made possible to setup the finite element model of a generic ride starting from Customer's requests like the ride layout and emotional response. The development of ride layout is in this sense managed entirely inside a 3-D CAD environment, so, the power of this approach is that the numerical FE model can follow real-time modifications of CAD layout. Acceleration and velocities, for example, as well as inertial response, are re-calculated after any geometrical modification to ride layout.

On the other side, this versatility has also economic benefits. It not only permits one to quickly explore in advance different ideas/scenarios that could satisfy the customer's desires, but also, as a



Fig. 1 - Loop 395 Coaster, side view of Finite Element Model



natural consequence, gives strong indications about the benefits of investments in general.

Besides, the ride must be subjected to strict rules and structural checks of all the components: track, columns and struts, basements, flanges... and cars as well!

In this sense, Loop 395 Coaster has been checked with the aid of a global finite element model, made mainly of beam elements type. Beam elements makes about the 90% of the ride structures, comprising rails, sleepers, diaphragms and so on...

Plate elements were used to model track panels. Track panels are very important and necessary to increase the stiffness of the track. In general, the layout of the track panels is not known before design, and has to be determined by practical and economic considerations. Their installation needs additional operations (say, welds) and costs, so, the challenge is to assume an 'optimal' layout of the panels.

Typically the procedure is: run FE model, check the results and determine if results satisfy structural requirements.

An important aspect of Roller Coaster design is deformation. It is surely true that you must have sufficient structural strength of a ride, but you also need a stiff structure in order to prevent large displacements of the tracks. In fact, when you are watching cars pass by from the outside the roller coaster, a rail that displaces 5mm seems more 'safe' than a rail that displaces 30mm!

The versatility of FE model can accommodate all these aspects. You have only to modify stiffness and re-run model.

Post-processing of FE models allows for tabular data like forces and stresses to be processed by a routine developed entirely inside EnginSoft in order to check structural members according to both the legal regulations and requirements of the customer.

The initial design, FE model and the structural checks of the Loop 395 Coaster were completed in a single month due to a number of

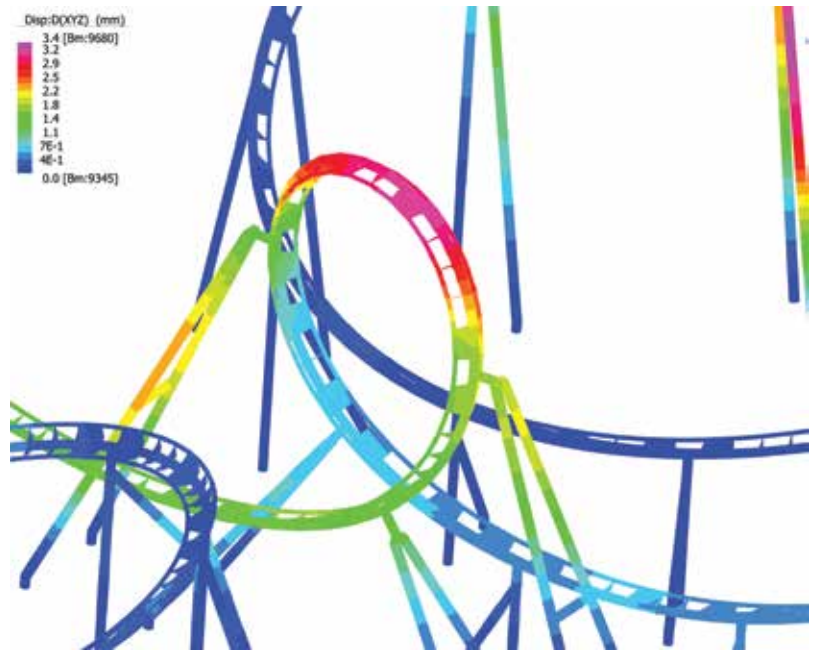


Fig. 2 - Displacement vector sum in the loop area

factors: IE-Park's high capacity of motivating the market and the final purchasers, Ride Tek Engineering's high level of skill and creativity in the design of Roller Coasters (we remember that RC's are not traditional steel structures and require high level technical expertise that are acquired only after years of hard work) and the analytical design competence of EnginSoft built through many years of projects with a wide variety of different customers.

So, we can say that the proper skills, specific competences, and, the passion for our work have brought us to the fulfillment of our challenging goal.

Ask the expert in Roller Coaster design:

Livio Furlan, EnginSoft

[l.furlan@enginsoft.it](mailto:l.furlan@enginsoft.it)



Fig. 3 - A ride manufactured by IE Park Loop area traveled by train

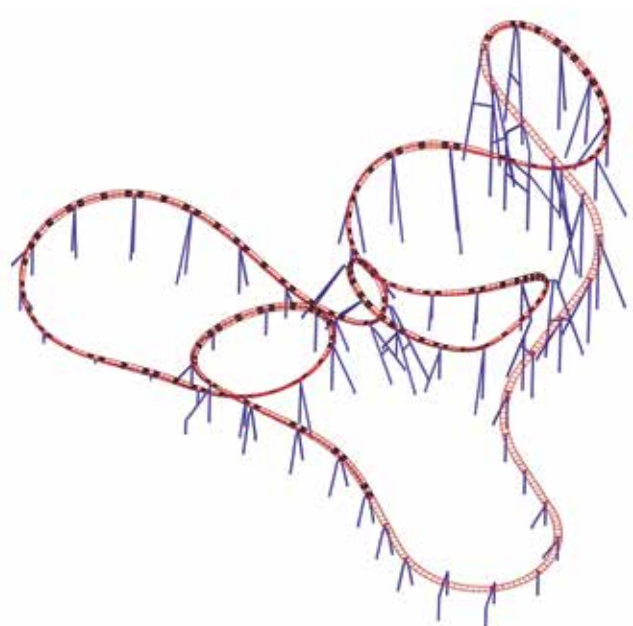
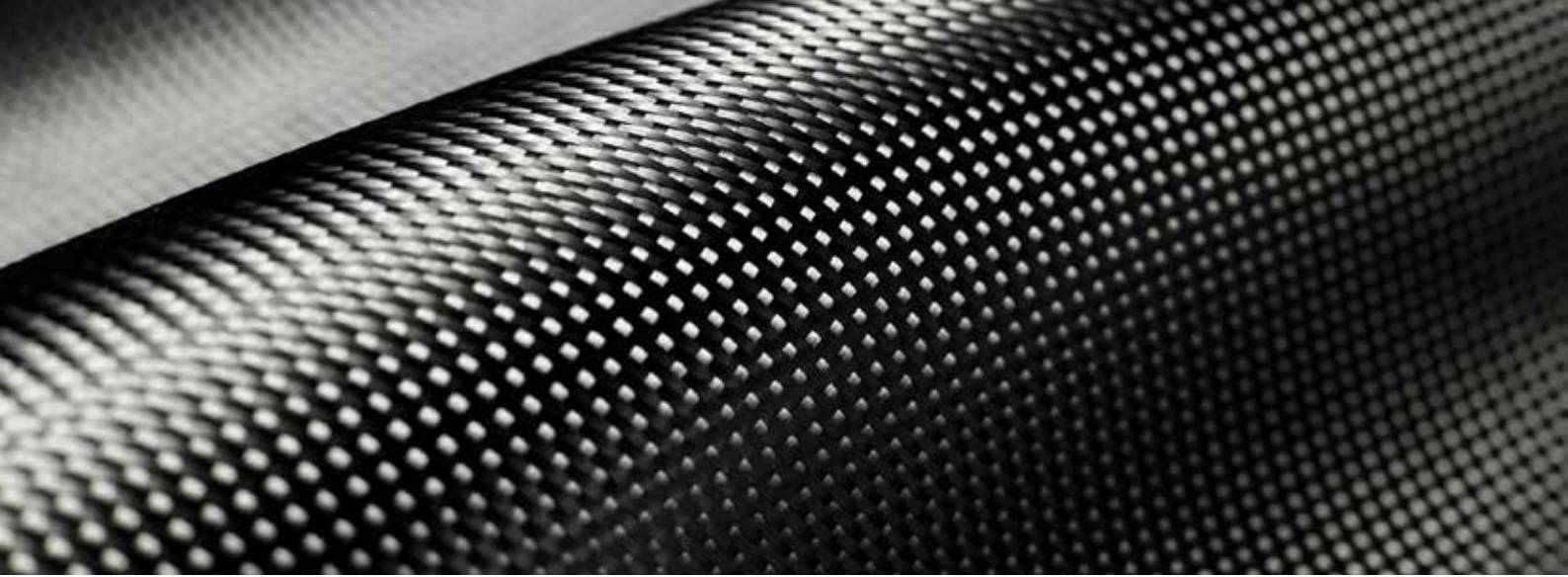


Fig. 4 - Loop 395, general layout of Finite Element Model



# Topological Optimization methods for aerospace applications, weight reduction of a RTM Composite RIB

## Topological optimization techniques, improvement of the strength/weight ratio on a RTM carbon rib

The evolution of the Computer Aided Optimization methods are frequently used in all product/process development phases. Some innovative optimization techniques, based on a topological approach, allow design improvement not only on the last phase for the choice of the best configuration, but also in the preliminary phases, sometimes providing significant and determining information for the structural performances optimization. From this point of view, the aerospace industry represents an excellent test bed: the possibility to modify the first attempt design, optimizing suitably the shape profiles and then setting up the manufacturing phases, it's an opportunity that provides new ideas and perspectives to get the desired design requirements.

EnginSoft and Compositi Avanzati in synergy applied their competencies on numerical simulations and production technologies for composite components, in order to develop a methodology that allows to identify the potential lightening



areas on a composite rib, placed within a wing box. The carbon-reinforced rib has been built through the Resin Transfer Molding technology, a closed-molding out-of-autoclave manufacturing process which guarantee high mechanical performances and relative low operative costs.

## Composites in Aerospace, comparison between autoclave and RTM production techniques

In the production of composite aircraft components, autoclave curing helps to achieve the desired fiber content and to minimize the voids in the resin. The high strength/mass ratio is achieved by placing vacuum conditions on the composite component within an autoclave and then pressurizing the autoclave during the heated cure cycle. Resin Transfer Molding (RTM) is a closed molding method alternatively used to fabricate performing

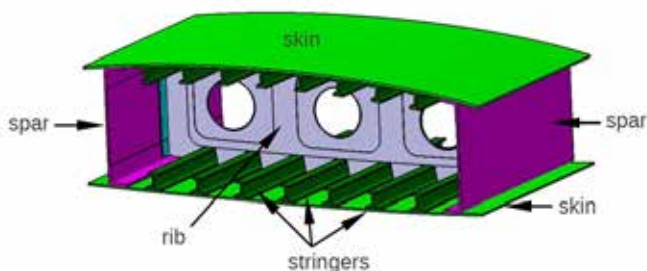


Fig. 1 – Section of the wing box

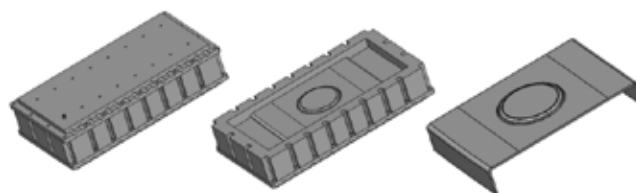


Fig. 2 – CAD geometries of the modular mould and rib



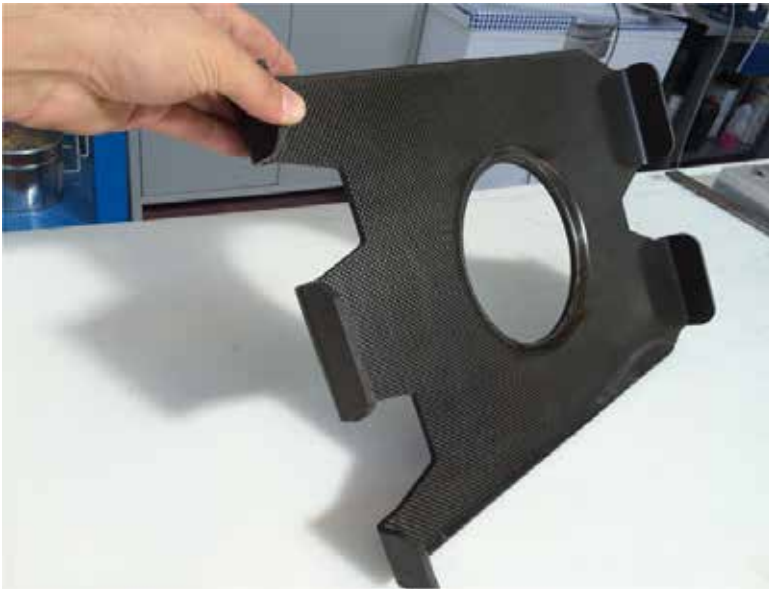


Fig. 3 – RTM carbon rib after machining operations

The “ANSYS Shape Optimization” module, used to perform the topological analysis, doesn’t automatically remove the low stressed elements; the new geometries are defined by the analyst, in parametric way if possible. The lightening areas are parameterized both in terms of positions and dimensions. According with the weight target, the low stressed areas are identified and removed through post-cured machining operations; after that, the exposed drilled or milled trough-the-thickness surfaces may be accurately covered with a coating in order to avoid the beginning of any delamination crack front.

Ask the expert in Composites:  
 Fabio Rossetti, EnginSoft  
[f.rossetti@enginsoft.it](mailto:f.rossetti@enginsoft.it)

composite structures according to quality typically required of aerospace applications. This process allows to realize composite components with more complex geometries, better dimensional tolerances, and same or higher strength to mass ratio compared to autoclave cured parts.

### Main properties of Resin Transfer Molding

RTM uses a closed mold commonly made of aluminum or other metal alloy. A fiber “layup” such as graphite is pre-formed and placed into the mold. The mold is closed, sealed, thermostatted and placed under vacuum, to extract any air and facilitate resin permeation. Resin is then injected into the mold up to 10 bar pressure to impregnate the fiber layup. The mold is then held at the appropriate temperature to cure the resin. Current RTM technology produces lightweight parts with excellent mechanical and aesthetic properties. With these qualities, composite materials are gaining wide use in a variety of structural and non-structural applications common in aerospace and aviation.

### Results and conclusions

The implementation of the topological optimization procedure allows us to identify a well-defined trend in the lightening areas; in fact, through a sensitivity analysis, it’s possible to find where the low stressed areas are located and remove them according to the mass reduction requirements.

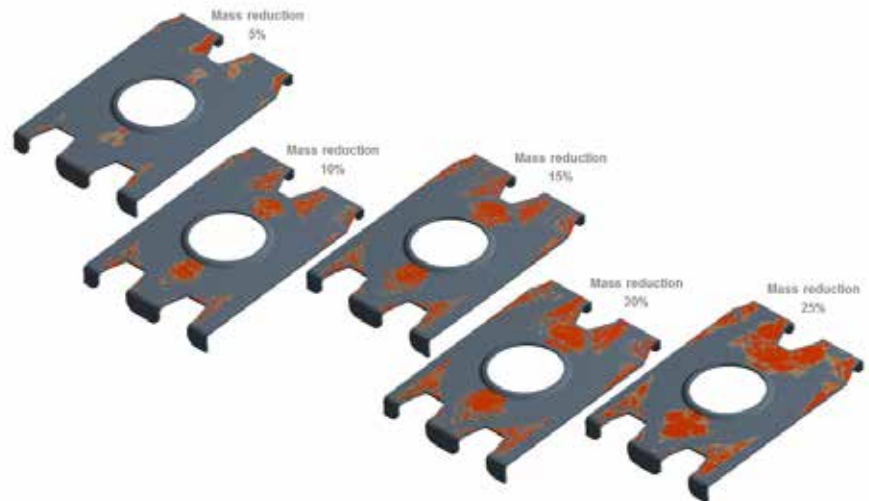


Fig. 4 – Results obtained by the topological optimization of the RTM carbon rib



Fig. 5 – Lightening of the RTM carbon rib exploiting the results of the topological optimization



# Optimization of a low noise hydraulic piston pump

## Minimize the noise emission while maintaining the same level of performance in a pump

The use of compact axial piston pumps of the swashplate type as input power source for modern hydraulic circuitry is common, nowadays, on both fixed and mobile applications. Beside the constant focus on pump efficiency and performance improvements, within the last few years significant attention has grown on pump noise reduction, especially for electrically powered systems as for example injection molding machines.

The objective of this project was to take a standard pump (MVP60, Figure 1) with a maximum displacement of about 84 cc/rev and to minimize the noise emission, while maintaining same levels of performance.

### New technology

It is well known that pump noise mainly comes from pressure fluctuations in the hydraulic circuit (fluid-borne noise) and large alternating forces inside the pump, both of which leading to the structure vibration (structure-borne noise) and the sound propagation on surroundings (air-borne noise, Figure 2).

On axial piston pumps of the swashplate type one solution that can virtually break down the flow ripple, and the pressure fluctuations in the hydraulic circuit arising from it, is the use of a pre-compression volume: it consists of an additional oil chamber able to pre-compress the fluid inside each piston before it is delivered to the pressure line.

In order to achieve the prospective benefits of this device, that are the reduction of backflows from the outlet line and the decrease of



Fig. 1 - MVP series axial piston pump

outlet flowrate peaks, a proper connection between components (Figure 3) must be designed to synchronize the external chamber recharging phase.

A key design feature for an effective result is a proper delivery timing, which is defined by the geometry of the pump valveplate: it decides when and how to connect each pumping piston to external ports and, further complicated, to the pre-compression chamber. In Casappa R&D Department one simulation tool able to predict the pump functioning is the 1D circuital model depicted in Figure 4, developed in LMS Imagine.Lab AMESim environment

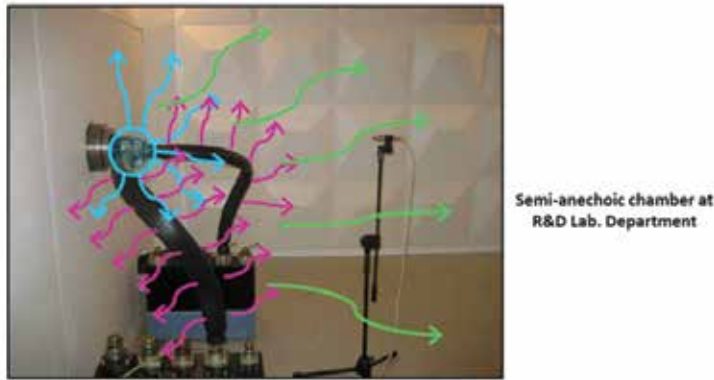


Fig. 2 - Sound propagation from the pump

in cooperation with the Polytechnic of Turin. This model estimates pump features related to noise as for example piston pressure peak, internal backflows, inlet and outlet flow ripple, cavitation conditions and oscillating forces between inner components.

### Optimization of the new pump

Once the AMESim model has been updated to consider the presence of the pre-compression chamber, a workflow inside modeFRONTIER software has been created (Figure 5) in order to optimize a new pump with this solution.

Two input variables define the pre-compression chamber volume and connection with outlet line, while other twelve variables fix the valveplate geometry; four constraints on inputs ensure that each configuration can be manufactured.

Two parallel AMESim nodes simulate the pump in five working conditions, combining different ranges of outlet pressure, rotational speed and displacement, in order to be sure that the pump will perform well for each utilization.

The simulation data are post-processed by an Octave script into twelve output variables, expressing the interesting pump performances listed before.

The most influent outputs (internal backflows and flow ripple) have their own objective function for a total amount of 5 OFs, while each other output is constrained according to the restrictive assumption that each feasible design has to be not worse than the current series pump.

A screening analysis of about thousand designs (Figure 6, 1st step) allowed to understand that the performance improvement is proportional to the volume of pre-compression chamber, while the connection with outlet line can be avoided because it slightly affects the OFs. The valveplate optimization, corresponding to the 2nd step, is composed of a DOE population of 120 designs (30 best configurations from previous simulation + 90 Sobol designs) and a standard Scheduler (MOGA II

algorithm, generational evolution, elitism enabled with 25 generations) for a total amount of 3000 designs and 5 days of simulation.

An iterative approach has been chosen, frequently checking and adapting the input variable ranges aiming to follow the optimization algorithm tendency. The final Design Table is composed of 5877 designs, of which only 119 are founded to be feasible.

The optimal solution has been chosen by means of bubble 4D charts (Figure

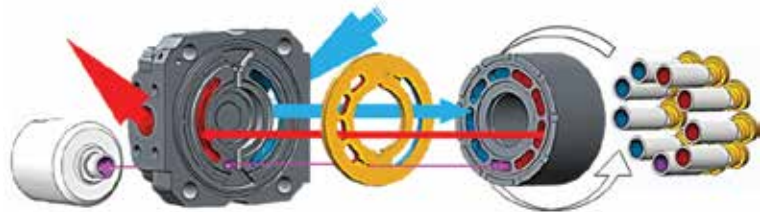


Fig. 3 - Connection between pistons, barrel, valveplate, cover and pre-compression chamber

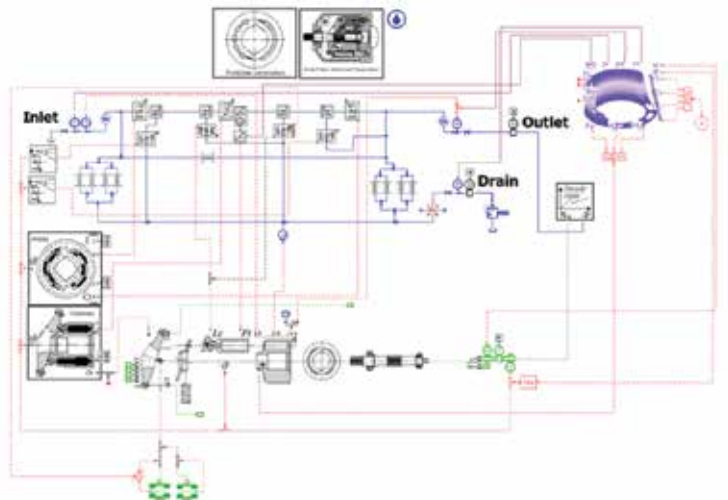


Fig. 4 - AMESim model of the MVP60 pump

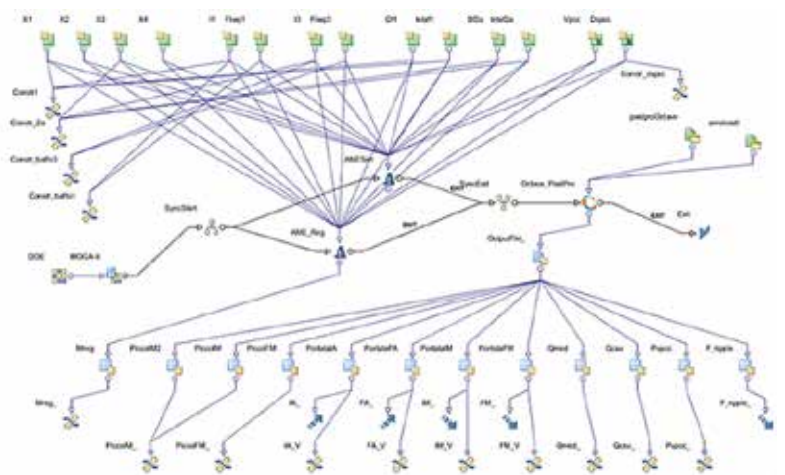


Fig. 5 - Optimization workflow



7) and parallel chart filtering: a reduction of 67% and 79% has been achieved on outlet flow ripple and internal backflows (OFs), while all other features have been improved or are unchanged (Figure 8).

In the optimization process it can be noticed that a 4th step was planned for the final refinement: once the optimization was over and the best design identified, a MOGA II has been run restricting the input variable ranges near the optimum combination; this additional step has led to further small improvements (Figure 8).

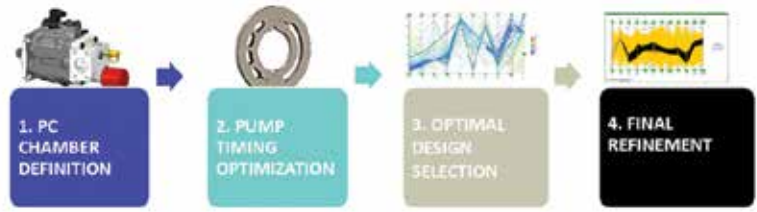


Fig. 6 - Optimization process

### Prototype and experimental tests

On the basis of the really promising results listed above, the new pump configuration has been prototyped and tested in our Lab Department.

The efficiency levels and cavitation are confirmed to be in line with the current series pump, whereas the pressure ripple highlights a significant lowering, with a maximum reduction on the 1st harmonic of about 71%.

Finally, the most important comparison, the noise emission has been measured in the Lab Department semi-anechoic room according to standard UNI ISO 4412-1:1994: a reduction of 3dB(A) on maximum values has been reached (Figure 9).

*Manuel Rigosi, Antonio Lettini  
Casappa*

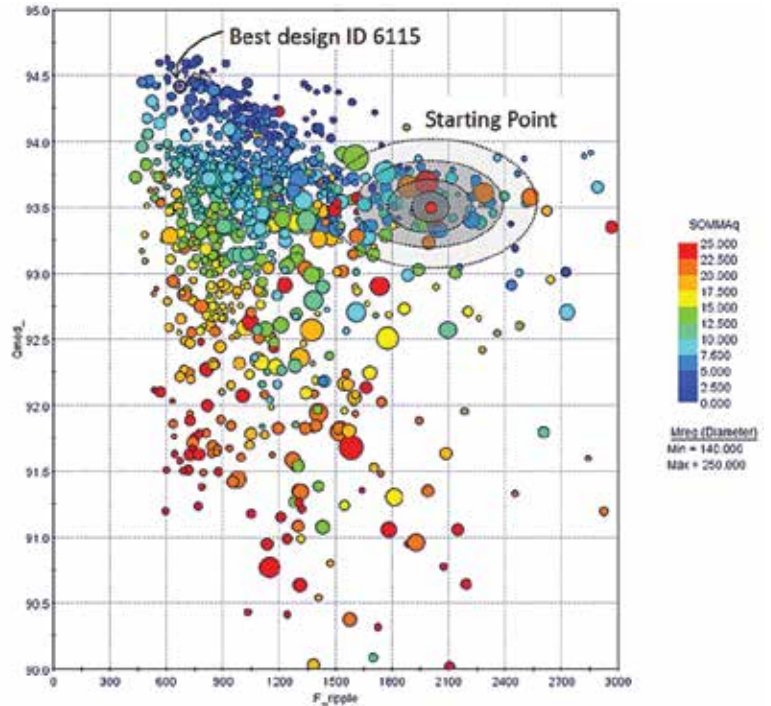
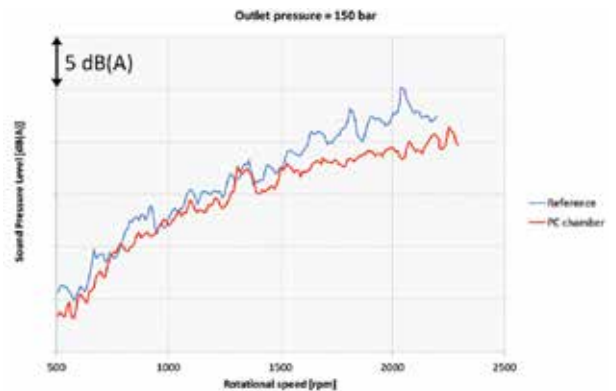


Fig. 7 - Optimal design selection

Ask the expert in oil piston optimization:  
Francesco Franchini, EnginSoft  
f.franchini@enginsoft.it







# ANSYS Mechanical customization through ACT extensions

## Some useful tools to improve and speed up the user everyday activities and ANSYS Mechanical features

The ANSYS Application Customization Toolkit (ACT) provides tools for customizing the ANSYS Mechanical product. ACT provides simple access to the main components of Mechanical, such as GUI components, mesh, geometry, boundary conditions, etc. in a Python programming environment. We have developed more than one extension to simplify and standardize many Mechanical actions. In this paper some examples of the ACT extensions we developed are described. They are used in everyday activities of SACMI Imola S.C. ceramic engineering department to improve and speed up the user operations. We propose here different types of ACT extensions: Encapsulation of MAPDL scripts (Bolt Verification), Creation of User-Defined Results (Radial Deformation, Shape Morphing), Encapsulation of .js Macros (Select External Faces) and other Miscellaneous operations (Free Viewer Translator, Utilities4Submodeling). A performance comparison in using or not ACT extensions is also provided.

### Introduction

ACT is a completely new and fully documented customization environment available since version 14.5 of ANSYS Workbench. It allows legacy ANSYS MAPDL expertise to be reused in order to produce straight-forward customized solutions for the ANSYS Workbench Mechanical environment. It allows definition of customized loads, solvers and results. Among the others, one



of ACT main advantages is the possibility to easily extend ANSYS Workbench capabilities towards the needing of single users. This paper shows some extensions we developed to help us during the every-day activities.

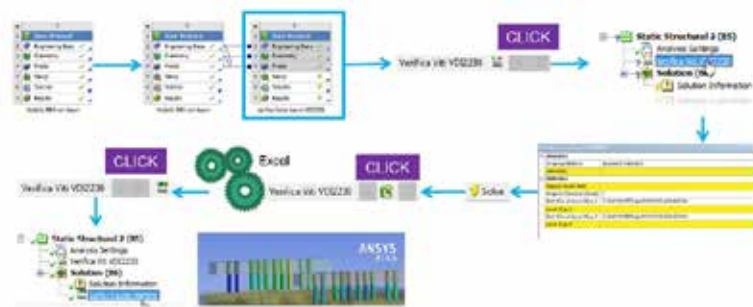


Figure 1: Bolt verification procedure by using ACT extension

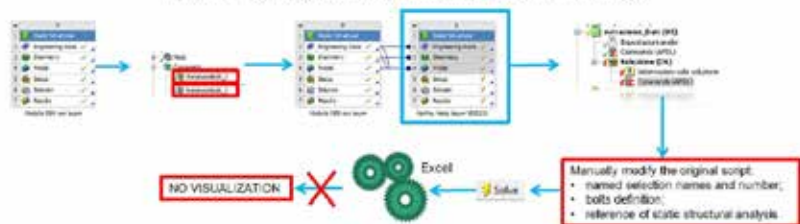


Fig. 2 - Bolt verification procedure without using ACT extension



Fig. 3 - Radial deformation without ACT



Fig. 4 - Radial deformation with ACT and examples of results

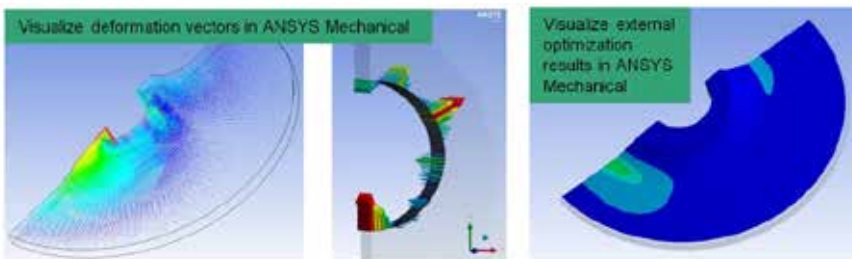


Fig. 5 - Shape Morphing results: ACT visualization

### Bolt Verification

Bolts are one of the most used components in mechanical applications and FEM analysis is the only way to determine their actual working conditions. Following the VDI2230 standard we use 1D beams to simulate bolts in ANSYS. In the past we developed a methodology based on APDL scripts and Microsoft Excel calculations to verify bolts of our machines. Thanks to ACT we migrate from APDL scripts to a Mechanical panel and we have now the possibility to visualize Excel results in Mechanical interface. Figure 1-2 show the different procedure of using ACT or not. The user selects the studied beams and applies a custom load instead of creating a named selection per each beam. The custom load reproduce the APDL script by writing in the FEM input file the information provided in the details panel. Excel can be open from ACT button and after the evaluation a custom result can be used to visualize results in Mechanical.

### Radial Deformation

In many mechanical applications designers need to know the deformation of a circle feature (e.g. seals, bearings, ...). Obtaining such an information from FEM is not a novel, but some additional post-processing steps are necessary and a dedicated tool inside Mechanical makes the user's activity much easier. The major difficult for the user is the splitting of rigid motion of the circle and its deformation. Before ACT we exported a circular path deformation

and evaluate rigid motion and deformation in Excel. Thanks to ACT the all calculation is made by Python and the results appear to the user in the Mechanical tree. In order to have a tool as general as possible we introduce symmetries and different results both for deformation and rigid motion. Figure 3-4 show the different procedure of using ACT or not.

### Shape Morphing

ACT can be used, in general, to visualize in Mechanical any kind of external results for advanced post-processing. Starting from a text file with desired results, it's possible to read the file thanks to Python capabilities in reading text, eventually elaborate data and finally plot them in Mechanical Graphic Window. Here we provide an example of such a capability by plotting in Mechanical the vectors of mesh deformation due to an external morphing tool. In particular we use modeFRONTIER to manage the shape optimization loop, ANSYS to compute FEA, Excel for fatigue analysis and Altair Hypermorph to define morphing rules. The ACT extension gives the user the possibility to see both the setup and the results of shape optimization inside Mechanical. Figure 5 shows the results.

### Select External Faces

Some users might be concerned by the fact that they would like to re-use their jScript macros in new ACT environment. We faced such a problem and we propose here an example of it. In particular we use few js functions to select and deselect faces in Graphic Window. The objective of this extension is the automatic selection of the all external faces of the model: it selects all the faces of the model and then deselect the faces that belong to more than one body. Such a tool become fundamental when the model is characterized by many faces and multibody part with many bodies. Figure 6 shows an example of the extension application.

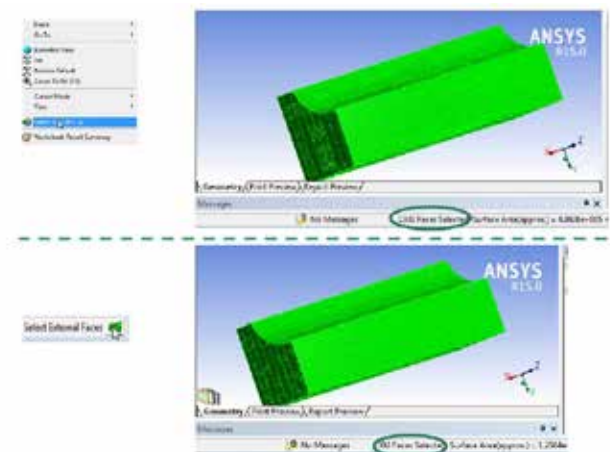


Fig. 6 - External faces extension: how it works



Fig. 7 - Python batch loop for result translation



Fig. 8 - Bodies IDs feature



Fig. 9 - Faster Submodeling feature

### Free Viewer Translator

The great flexibility and powerful of Python language can be used in ANSYS ACT extensions. In the current example we provide a demonstration of it. We developed a methodology to generate light results file readable by a free viewer in order to share results among different departments of the company. The full methodology is managed by Python and can be summarized as follows: the result file is copied in a temporary folder, Python executes in batch the software we use to create light result files (i.e. HyperTrans and HyperView) together with dedicated macros for each kind of result. At the end of the translation Python cleans the temporary folder and sends an e-mail to the analyst with the path of new light results. This extension appears to the user as a single button which has to be clicked at the end of the analysis while Python works in batch even after the Workbench shut down.

### Utilities 4 Submodeling

Submodeling is a well-known finite element technique that the analyst can use to obtain more accurate results in a particular region of a model. It let the user to use a coarse mesh in the Entire Model (EM) and refine it in a new independent model of only the region (SubModel (SM)) of interest and then analyze it. Such a powerful technique is integrated in ANSYS Workbench since release 14.5 giving the users lots of possibilities and advantages. However, our extension adds three small features which seem to be necessary for our daily use of Submodeling.

It is known that the EM can be both an assembly or a part. If it is an assembly, the interpolation scheme can wrongly take the displacements from different parts of the EM. Workbench

Submodeling allows the analyst to manually choose the EM bodies to be considered in interpolation but the proposed way appears to be difficult to handle in some cases. In fact, the user should get the matIDs of the needed bodies from the output file of the EM analysis. Such an operation could be time-consuming and bring to errors if the EM is large and complex. The first feature of the proposed extension does it for the user. A custom result called "BodiesIDs" is available in the EM. By applying it to right bodies, Python opens (through notepad) a txt file where the user can copy (and paste to SM) the requested IDs. The selected bodies are also colored with corresponding IDs.

The second feature of the present extension is useful when the SM is linear and the user wants to speed up its solution. By clicking the second button of the toolbar, Analysis settings are modified and an (eventually editable) APDL script is added to the Mechanical tree.

The third feature of the present extension is necessary especially when the EM is large and the user storage is finite. Each time a new SM is created, the .rst file/s of the EM is copied in a new folder (called "Solution ##") inside

the "MECH" folder of the SM. Such a file is necessary for solving the SM but after that it becomes redundant. Our extension simply delete that file/s when the user click on the third button of the "Utilities 4 Submodeling" toolbar.

### Conclusions

ACT appears to be a powerful tool for users who want to customize ANSYS Mechanical features (it works for Design Modeler as well). All the described extensions have been developed with some scripting skills. Programmer skills are necessary for more advanced extensions but usually they are not. As long as a CAE platform wants to be close to CAE users, customization tools are necessary and ACT is the right way forward.

Riccardo Cenni, SACMI Imola

Ask the expert in ANSYS ACT Customization:

Francesco Micchetti, EnginSoft  
f.micchetti@enginsoft.it

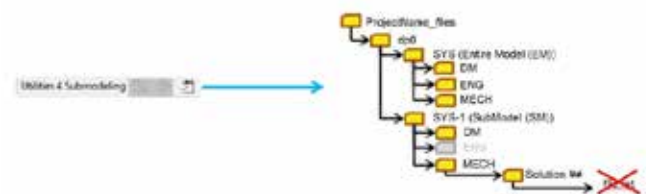


Fig. 10 - Clean redundant RST feature





# Topological Optimization procedure for civil steel frame design

## Introduction

In the Civil Engineering industry, design optimization of structural components is often obtained through methods applied on already existing solutions. These methods are based on a “library” approach which allows the selection of materials and dimensions, overlooking where they are located.

An optimization procedure coupling FE ANSYS Workbench with modeFRONTIER is presented here: The aim of the work is to optimize the beam’s dimension and location within a steel frame structure, fully modeled through FE beam elements and joint connections; its design and mechanical performances can be compared with other “standard” configurations.



## Topological optimization procedure

The numerical optimization process is a semi-automatic procedure composed of different phases:

- double preliminary topological optimizations of the trusses (first phase) and the roof’ support (second phase);
- definition of the optimized geometry on the base of the results obtained by topological analyses;
- assessment of the static and dynamic structural behavior;
- multiobjective optimization of the whole parametric system.

The technical intervention of the analyst is required after the preliminary optimizations, in order to interpret the suggestions provided by the topological study and to define the most suitable design in terms of technological feasibility.

## Description of the FE model

The load case in question is the design of a steel frame structure for a warehouse, composed of four trusses 15 m large, 7 m tall and 33 m deep. The dummy structure has been built with 2D/3D elements; once the FE boundary conditions were applied, the “Shape Optimization” tool in ANSYS allowed the identification of elements to be kept or removed on the basis of mass reduction. After a qualitative analysis of the suggestions obtained by the topological analyses, the analyst has to define the new optimized

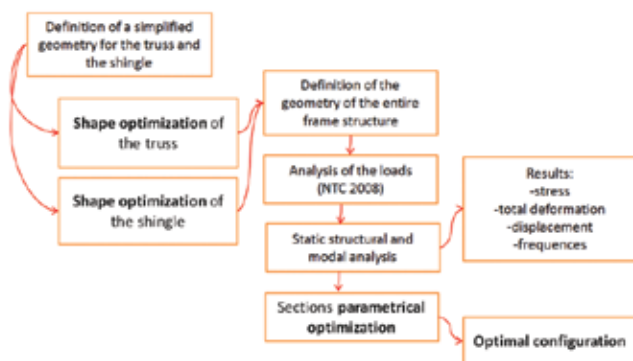


Fig. 1 – Optimization process logic

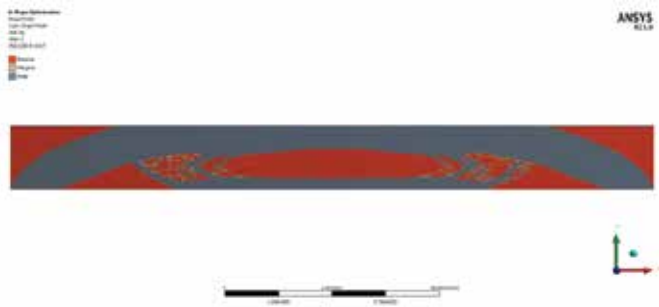


Fig. 2 – Shape finder – Target reduction of the truss

geometric configuration of the whole system. The FE model of the steel structure is built through Beam and Joint elements; its mechanical behavior is analyzed under the applications of several load conditions, carried out according to the Standard Rule NTC2008.

Finally, a parametrical optimization of the beam elements' sections is performed in order to improve the following objective functions:

- mass minimization;
- maximum displacement minimization;
- maximum equivalent stress minimization;
- first natural frequency maximization, according to the earthquakes requirements.

### modeFRONTIER multiobjective optimization of the steel frame

A genetic algorithm is used to determine the section's optimal dimensions. A "library-based" approach with a vector parameterization method is implemented to choose the dimensions of the rectangular and IPE sections.

This entire multi-step optimization procedure achieves the best configuration of the whole steel structure, both in terms of design and mechanical performances. In detail, the final truss is represented by two arches lightened on the low stressed regions. According to the topological results; the support beams for the roof are arranged in the most simple and effective way. In addition, the best shape and size of the beam' sections are identified.

The procedure of optimization described above is a cutting-edge method of optimization: The analyst is able to find the optimized configuration for any structure, both new and existing, improving the final design performance.

Carmen Bernardini – University of Salento  
 Giorgio Zavarise – University of Salento  
 Fabio Rossetti – EnginSoft S.p.A.

Ask the expert in Topological Optimization:  
 Fabio Rossetti, EnginSoft  
 f.rossetti@enginsoft.it

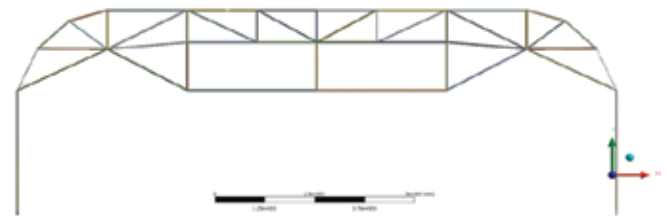


Fig. 3 – FE model of the truss – Front view

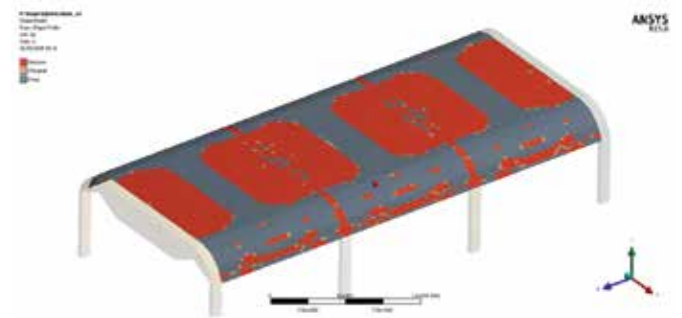


Fig. 4 – Shape finder – Target reduction of the roof' support

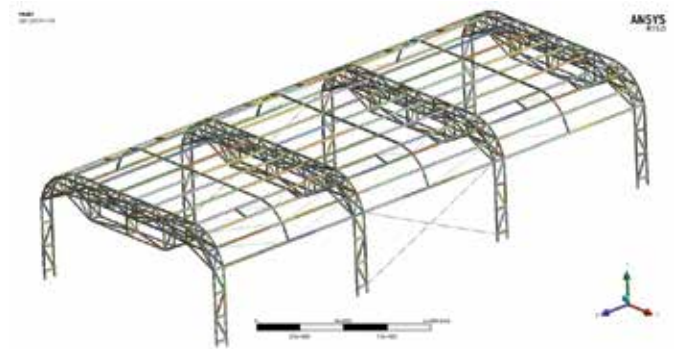


Fig. 5 – Final configuration of the steel structure

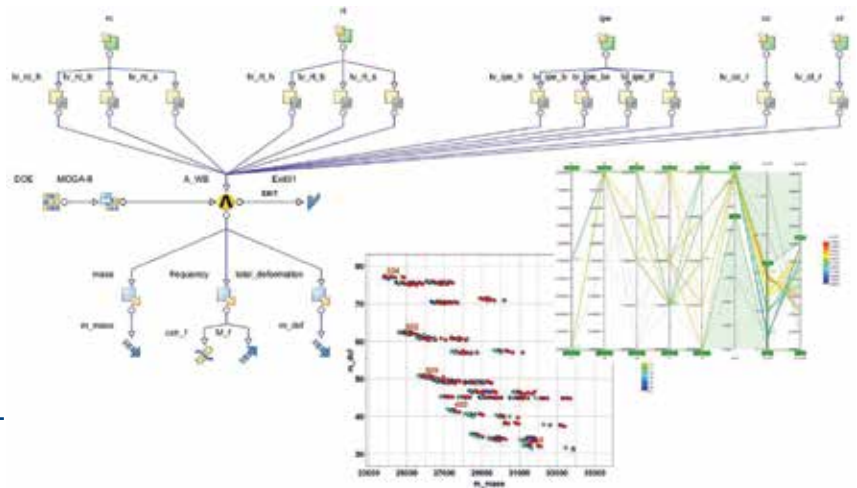
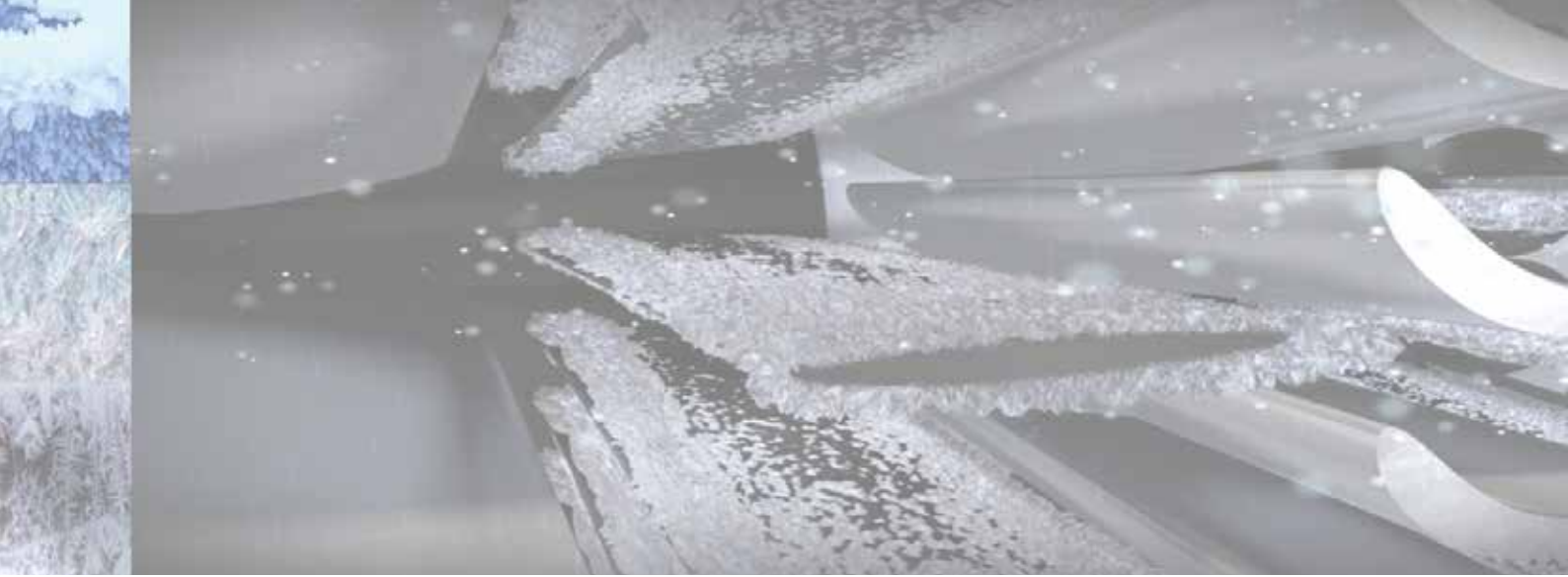


Fig.6 - modeFRONTIER optimization workflow and post-processing charts



# Computational Fluid Dynamics prevents engine damage in Ice Crystals conditions

Ingestion of Ice Crystals (IC) at high altitude can cause engine damage and power loss. With recent advancements in Computational Fluid Dynamics (CFD), such adverse effects can be predicted and avoided

In the last 10 years, a particular icing phenomenon has caused more than 100 aviation incidents at cruise in high altitudes. Such incidents involved aircraft engines that suffered from either a loss of power, combustor flameouts, complete failure due to compressor surge/stall, blade damage or excessive vibrations. These failures are thought to have been caused by Ice Crystals ingestion.

If ice accretion on the compressor blades is large, the resulting ice shapes can break off due to aerodynamic forces and shed into the combustor. This can cause an engine flameout and will require an engine restart. In more serious scenarios, a compressor surge/stall can occur, leading to rapid and catastrophic breakdown of the engine.

## What are Ice Crystals (IC)?

IC are generally formed as a consequence of natural convection within deep convective clouds, such as Cumulonimbus clouds shown in Figure 1. The movement of liquid water droplets from the bottom of these clouds to the top, combined with the rapid decrease in temperatures below freezing experienced by the drops during their ascent, causes the formation of IC

## Ice accretion effects

IC can be a serious threat to engines. When ingested by an engine's compressor, the crystals experience a higher temperature. The crystals eventually may melt through the compressor stages, and stick to internal surfaces. Surface temperatures drop due to evaporation and eventually allow ice accretion. The components most affected are the stationary parts, such as stators and Inlet Guide Vanes (IGV).

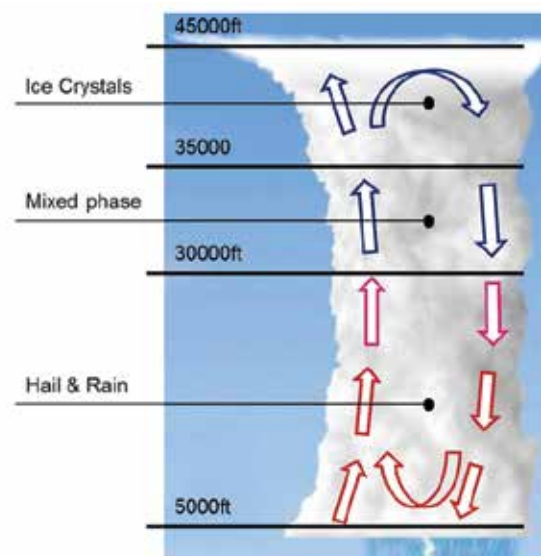


Fig. 1 - Schematic representation of cumulonimbus cloud



### Simulation of IC effects

The impingement of IC on engine internal surfaces and the subsequent ice accretion are modelled using Computational Fluid Dynamics. Mathematical models determine the impingement and ice accretion characteristics of IC. Compared to experimental testing, the CFD approach is a fast and cheap methodology and allows the assessment of an engine under all atmospheric conditions. The process for simulating ice accretion is summarized in Figure 2.



Fig. 2 - Computational process to determine ice accretion

First, CFD codes (Figure 3) compute the flow field inside the engine, the forces acting on the blades as well as the heat transfer. Using the airflow solutions, the ingestion of crystals by the engine is analyzed and their locations of impingement are determined as shown in Figure 4. Zones where there is a higher collection of IC will experience higher ice accretion.

Several models can be applied to study the interaction between the crystals and the engine's internal surfaces. They cover all possible flight conditions, crystal shape and sizes, as well as bouncing and shattering. The simulation method is fully compliant with the recently announced (November 4, 2014) FAA aviation authority's regulations.

With the information provided by the flow and impingement solutions it is then possible to compute ice accretion due to IC, shown in Figure 5.

### FENSAP-ICE-TURBO

One of the most widely used ice accretion simulation software specifically designed to analyze jet engine icing is FENSAP-ICE-TURBO, developed by Newmerical Technologies International. FENSAP-ICE-TURBO is tailored to the needs of jet engine companies, and is capable of simulating the entire spectrum of Appendix C, D and O for droplets, ice crystals and SLD environments. Blade rows are coupled by exit/inlet boundaries, each pair forming an interface where data is exchanged through a pitch-averaging algorithm. There is no limit to the number of blade



Fig. 4 - Ice Crystal impingement zones on engine surfaces

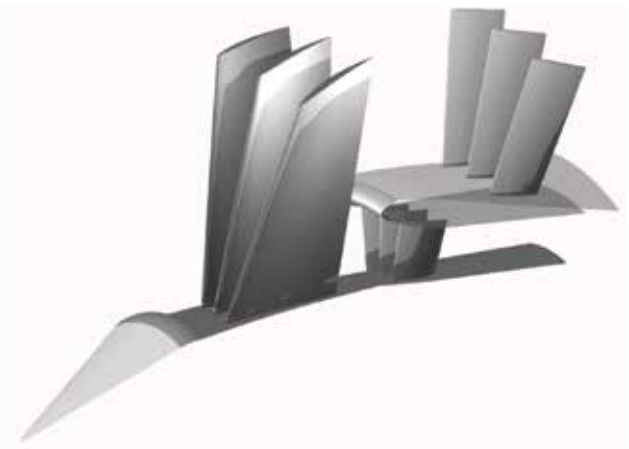


Fig. 5 - Ice accretion on a Turbopan

rows that can be linked, nor on the type of linkage. Sequential or staggered arrangements are possible.

An advanced Graphical User Interface (GUI) links the four modules seamlessly. Each module is compatible with unstructured and hybrid grids (hexahedral, tetrahedral, pyramid and prism elements), the same grid being shared by all the modules during the analysis process.

Compatibility with other commercial CFD codes enhances productivity by lowering the incremental cost of any icing analyses. Once the ice accretion simulation is performed, it is possible to assess the performance loss caused by icing, and determine if the engine can still provide acceptable thrust to the aircraft. FENSAP-ICE-TURBO can also aid in the design and evaluation of engine Ice Protection Systems.

*Elia Cipolato, Massimo Galbiati*

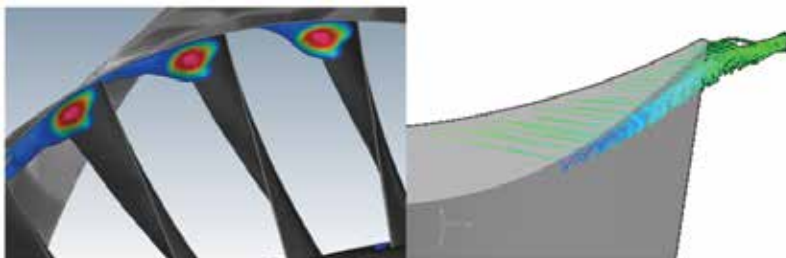


Fig. 3 - Tip gap vortices behind the fan (left) and compressor blade (right)

Ask the expert in Fensap-ICE  
Elia Cipolato, EnginSoft  
[e.cipolato@enginsoft.it](mailto:e.cipolato@enginsoft.it)



# Multi-Objective Optimization Analysis for Selection of Eco-Friendly Materials According to Itaca Protocol for Zero Energy Buildings in the Mediterranean Climate

The recent worldwide environmental issues impose to reduce the energy consumption and the greenhouse gas emissions of the building sector, keeping in mind the whole life cycle of construction materials. The Itaca protocol (Institute for Innovation and Transparency of Contracts and Environmental Sustainability) promotes the use of recycled, renewable and locally sourced materials; in particular, the definition of product's sustainability is complex, and the presence of one or more "eco" features does not necessarily make it "eco" in its entirety.

A multi-objective analysis has been carried out in order to identify high energy efficiency external walls for Zero Energy Buildings (ZEBs) in the warm climate, privileging eco-friendly building materials. The definition of the external walls for Mediterranean climate with an optimal multilayer package through the integration of a multi-criteria optimization analysis was carried out in modeFRONTIER rel.4.3 environment with calculation procedures to evaluate the dynamic performance of building components developed in MatLab rel.7.0 environment. The optimization has been performed in terms of static transmittance, periodic thermal transmittance, decrement factor, time shift, areal heat capacity, thermal admittance, surface mass, thickness and Itaca score.

The topic of this paper is to define a method for the design of high efficiency external walls of new low-cost residential buildings among which the designer can choose the proper solution for his application, according to the Pareto front of the multi-objective problem.

## 1. Introduction

Several authors show how the construction sector consumes approximately 40% of natural resources, uses almost 12% of potable water and 70% of electricity. Construction activities can be more sustainable with appropriate strategies and actions; all the phases of a building's life cycle,



construction, fit-out, operation and ultimate demolition, have an important impact on human life and the environment. The impact of the construction process can be reduced by the use of the best eco-innovation techniques available on production processes, preferably locally produced. The building materials selection plays a fundamental role in the achievement of the 'Green Buildings' target and it is performed during the early stage of the design process, when the strategic choices concerning the building are made, and then in the working plan, when the materials available on the market are selected.

By the use of calculation procedures developed in Matlab, the modeFRONTIER optimization tool rel.4.3.0 was used to evaluate the dynamic performance of building components, in terms of thermal characteristics and the combination of several eco-friendly and locally sourced materials allows to maximize performance in agreement with the ITACA Protocol. The final aim of this study is to obtain a set of high efficient external walls, for buildings designed in the Mediterranean climate; it can be considered a tool supporting designers during the selection of materials.

## 2. ITACA environmental protocol

Although the availability of various information sources on sustainable materials is increasing, researchers have not agreed upon a clear definition of the term and its usage. Sustainable materials are materials with high recycled content, rapid renewable periods, high reused content, low-emitting contaminants. Sustainable materials are low consuming,

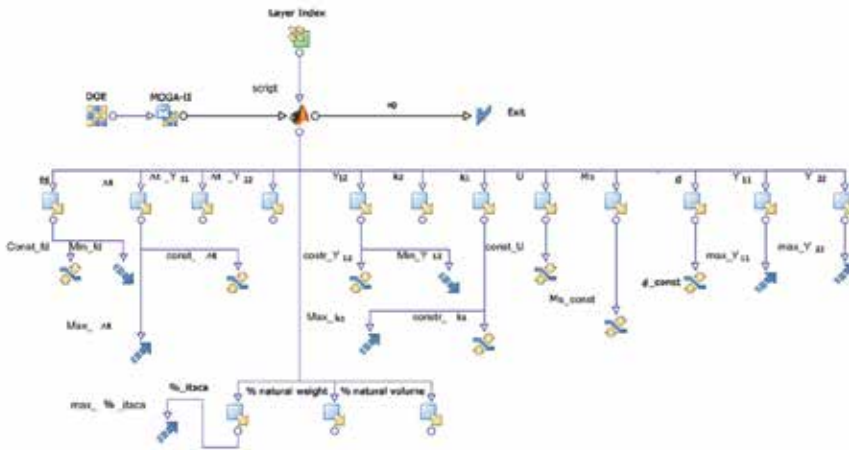


Fig. 1 - modeFRONTIER rel.4.3, flow-chart of the multidisciplinary and multi-objective analysis

low repairable and highly prolonged, easy to build with, safe to use and highly satisfying to the user.

The ITACA Protocol is a national assessment system for environmental sustainability. The principle is to share a common standard and valid at international level, as identified in the Sustainable Building Method (SB Method), but also to allow more configurations of it at local/regional level. The ITACA Protocol suggests the qualities of “eco-friendly building materials”: not toxic, reusable, renewable, recycled and locally sourced.

### 3. Cold and warm climates

Evola et al. and Voss et al. show that over the last 20 years more than 350 buildings, have been built to obtain a zero energy balance in all parts of the world. The study about ZEBs design in warm climate is not satisfactory; most of them are located in areas characterized by extremely cold climate. In warm climate, where the thermal overload is irreversible when radiation is not controlled and the free supply of heat indoors is mismanaged, it is necessary to study the thermal mass and the thermal inertia of the envelope to achieve a good internal comfort in summer seasons.

Sami et al. show that high energy efficiency buildings build where the climate is cold, use the technology of multi – layered walls. Structural materials of low density and thermal isolation are used to achieve very low steady thermal transmittance. Congedo et al. studied how a hyper-insulating envelope does not allow the discharge of accumulated heat at night because of low thermal mass and low thermal inertia, thus different solutions have to be carried out in warm climates. The heavy building, in the Mediterranean area, has intrinsically useful features that modulate the complex and unpredictable daily events that characterize these warm climates. Sami et al. show the effects of varying amount and location of thermal mass on dynamic heat-transfer characteristics of insulated building external walls with same thermal resistance.

The EN ISO 13786 has been used for the calculation of the thermal behavior of building and the data required are thermal conductivity, specific heat capacity, density and thickness of each layer of homogeneous material. Generally, it is observed that an increase in the thickness of the wall will entail a rise in the time shift with which the oscillation peaks will occur, while an increase in the thermal aerial capacities will involve an increase in the decrement factor, that is, a reduction in the amplitude of periodic oscillations. In summertime, when high external temperature variations

occur (sun-air) it is required the wall to be able to considerably reduce and delay temperature peaks on the internal surface of the wall so that the thermal flow lost internally is diminished.

If the whole process is to be efficient, the time shift must be such that the temperature peaks should be brought on the internal surface at night time, when outside air temperature is lower. Low values of the decrement factor, jointly with high values of inside thermal areal capacity, and high values in the time shift of periodic thermal transmittance, indicate better characteristics of the walls when it comes to diminishing the effects of external thermal stress during summertime.

### 4. Methodology

The multi-objective optimization analysis was carried out in modeFRONTIER rel.4.3 environment with calculation procedures developed in MatLab rel.7.0 environment. This choice reduces the complexity of modeFRONTIER block diagram and allows a virtually instantaneous speed of execution of the single design in order to reduce the duration of the entire simulation. Fig. 1 shows the flow-chart of the multi criteria analysis. The process variables are attached to the thermal properties, thickness and eco-sustainability of each layer. The set of initial combinations are processed according by MOGA II (Multi Objectives Genetic Algorithm).

#### 4.1. Input variables

The configurations with five layers and a maximum total thickness of 430 mm have been preferred. The first step of the analysis is the definition of the vector input variable properties: in particular, the random choice of building material for each layer of the component, their thermophysical properties and commercial thicknesses.

An investigation about the entire life cycle of a set of building materials available on the market today has been conducted, and at each of them was assigned a rating referred to following points: recycled and / or recovered materials, materials coming from renewable sources, heavy materials locally produced, materials for finishes and eco-friendly materials.

The analysis is conducted in the summertime, the wintertime is not critic in Mediterranean area.

#### 4.2. Objective functions and constraints

Table 1 shows the different constraints and objectives. They are the result of design choices mainly related to dynamic thermal properties, weight and sustainability.

Two groups of walls have been considered, in terms of surface mass and internal areal heat capacity: Heavy ( $M_s > 300 \text{ kg/m}^2$  and  $1 > 80 \text{ kJ/m}^2\text{K}$ ) and Light external walls ( $M_s < 230 \text{ kg/m}^2$  and  $1 > 60 \text{ kJ/m}^2\text{K}$ ). The sustainability criteria was implemented using a single index (eco-friendly score) equal to the sum of singles scores of the subsections.

The first simulation step for each wall type yields a DOE (Design Of Experiments) sequence composed by 10000 individuals. These combinations have been processed by MOGA II imposing a number of generations equal to 10, for a total of 100000 reiterations. The MOGA II starts from a number of possible solutions (DOE), called “initial population”, and provides them evolve during the execution. At each



Output	Objectives	Constraints	Type
fd	MIN	< 0,11	-
$\Delta t$	MAX	< 20 h	-
$Y_{12}$	MIN	< 0,04 W/m <sup>2</sup> K	-
$\kappa_1$	MAX	> 80 J/m <sup>2</sup> K	Heavy
$\kappa_1$	MAX	> 60 J/m <sup>2</sup> K	Medium
$\kappa_1$	MAX	> 60 J/m <sup>2</sup> K	Light
U	-	< 0,4 W/m <sup>2</sup> K	-
d	-	< 0,43 m	-
$M_4$	-	> 300 Kg/m <sup>2</sup>	Heavy
$M_4$	-	[230-300] Kg/m <sup>2</sup>	Medium
$M_4$	-	< 230 Kg/m <sup>2</sup>	Light
$Y_{11}$	MAX	-	-
$Y_{22}$	MAX	-	-
% Itaca	MAX	-	-

Table 1 - Objective functions and constraints of multi-criteria optimization analysis

iteration, it makes a selection of individuals of the current population, using them to generate new elements of the same population, which will replace an equal number of individuals already present and, thereby, constituting a new population for the next iteration. This succession of generations evolves toward an optimal solution (local or global) to the assigned problem. The result of each simulation is a set of feasible solutions which represent the so-called Pareto front, these solutions are the best compromise between the various designated objectives and all the constraints imposed on the simulation model. The main goal is to have external walls with the following specifications: high areal heat capacity, extremely low periodic thermal transmittance, very low decrement factor, time shift in the range h, high eco-friendly score. At the end of the first step of generations the pareto solutions have been selected, like new initial population, for the second step of one hundred generations. At the end of the multi-objective analysis, the Pareto population have 2014 and 6063 individuals for light and heavy walls, respectively.

For the selection of the best configurations, the steady thermal transmittance was not priority; the first selection has been done in function of the eco-friendly score. Several configurations (about 6000, 1300 individuals for heavy and light walls, respectively) respected the maximum value of 88.8%. The second criterion was the highest values of  $\kappa_1$ . Between last selection, the solutions with the highest values of  $\kappa_2$  has been chosen. In the end, the choice was in terms of time shift in the range 12-18 hours, and lowest values of the decrement factor.

### 5. Calculation results and discussions

The results show that it is always possible to reach high performance external walls with limited thicknesses, not wider than about forty centimeters, using eco-friendly building materials currently available on the market, and achieving very high dynamic thermal performances for warm climates.

It has been used the correlation matrix, resulting

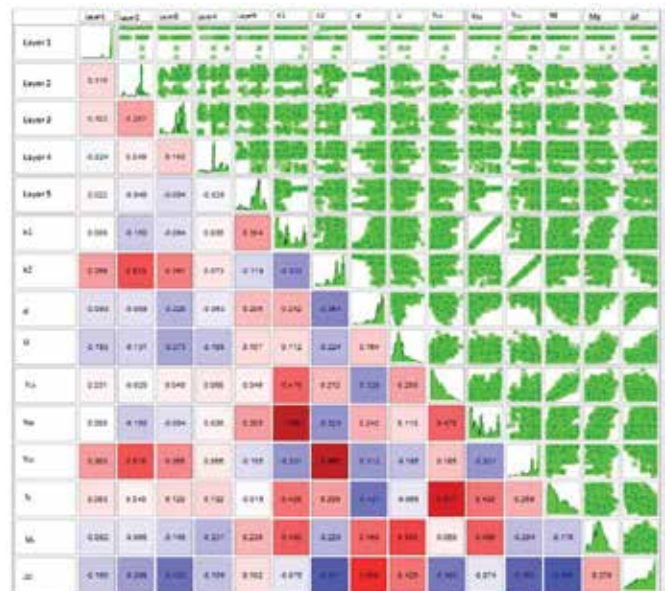


Fig. 2 - Example of correlation matrix for heavy external wall

from modeFRONTIER rel.4.3, Fig.2 shows an example for heavy walls. It denotes the degree of dependence between two variables. When the value is positive, the two variables have a direct proportionality, instead a negative value suggests an inverse relationship. The interaction is much more accentuated as the value is closer to 1 for the direct proportionality and -1 for inverse proportionality. When the correlation value is nearly zero there aren't relationship between the variables.

The histograms in figs. 3 and 4 put in evidence the presence of a particular building material type on a given layer for heavy and light external walls, respectively.

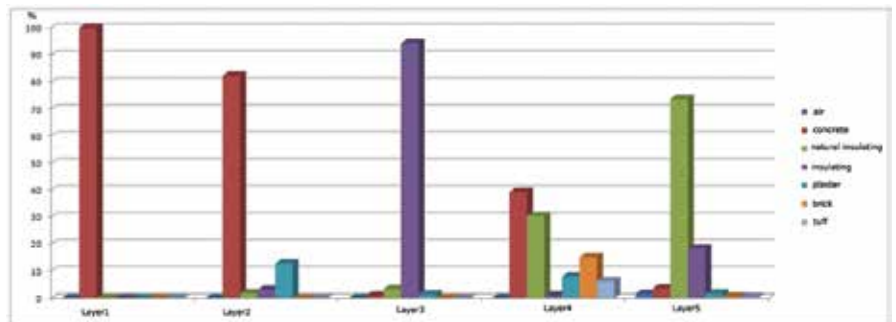


Fig. 3 - Histograms of probability of finding a building material on a given layer for heavy external walls

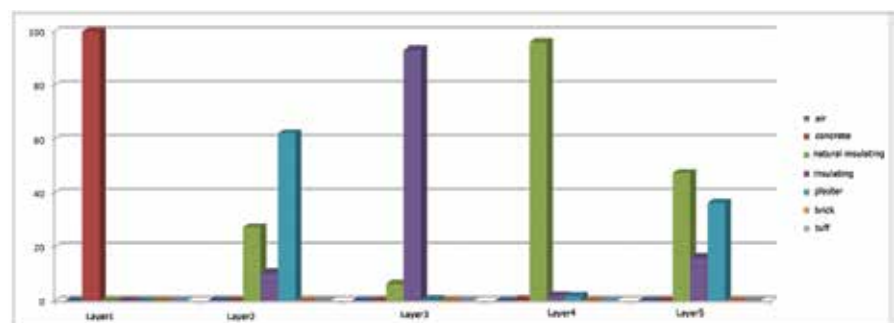


Fig. 4 - Histograms of probability of finding a building material on a given layer for light external walls

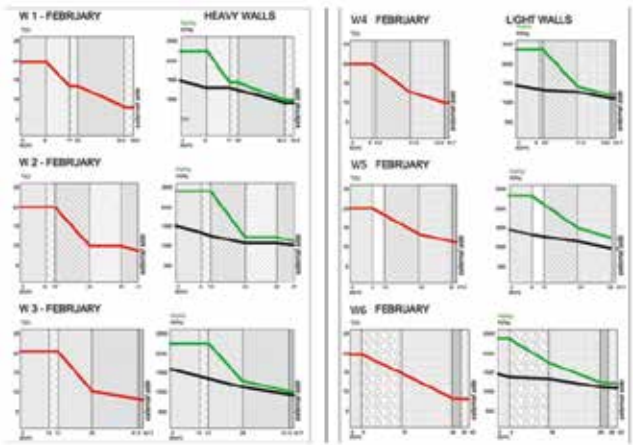


Fig. 5 - Heavy and light external walls - examples of high performance walls (Glaser check at February, Lecce - Italy).

The probability of finding a building material on a given layer for heavy external walls has been represented by histograms. Looking at fig. 3 for the heavy walls, there is a complete prevalence of concrete in the first two layers. The insulating material is completely prevailing in the third layers, regardless of the surface mass. Besides the natural insulating materials prevails in the fifth layer.

Looking at fig. 4 for the light walls. On the first layer, concrete is confirmed like the best material to optimize value of  $\kappa_1$ . In particular, for warm climate, also varying the superficial mass of wall, first layer with high thermal capacity is strategic on the internal side in order to obtain good performances. In the third layer insulating materials prevail. On the following layers natural insulating materials overcome.

In fig. 5 and in Table 2 and Table 3 a selection of the best configurations are represented, respectively for heavy and light weight walls. In agreement with the norm EN ISO 13788, the Hygrothermal performance test (Glaser) has been done. All configurations present a high ITACA Protocol score.

The observation of the best configurations suggest to move layer with high internal capacity toward the inner side and an insulating coat toward the outer side. The insulating coat can be a sequence of different materials to optimize the final performance. Concrete is the most common material that gives high thermal mass in buildings. Heavy external walls selected by pareto analysis show internal areal heat capacity greater than  $80 \text{ J/m}^2\text{K}$ , all solutions are composed at least by two concrete layers; in other words, because the concrete layers are subsequent, it is possible to realize heavy external wall with just four layers.

## 6. Conclusions

The different climates influence the definitions of the parameters for the design of zero energy buildings. In Mediterranean climate it is important to focus the attention on the thermal mass and the thermal inertia of the envelope in a temperate climate.

This study permits to simplify the analysis and to identify a simple external wall configuration in terms of cost and easy assembling on the building site ZEBs in warm climate, using ecofriendly materials. The selection of construction materials is considered as a multi-criteria decision problem, the designer can choose the proper solution for his work between a set of possible external wall configurations obtained according to the Pareto front of the multi-objective problem.

Cristina Baglivo, Paolo Maria Congedo,  
Andrea Fazio, Domenico Laforgia

Department of Engineering for Innovation, University of Salento

Ask the expert in optimization analysis:

Francesco Franchini, EnginSoft  
f.franchini@enginsoft.it

WALL	W1	W2	W3
Layer1	Concrete	Concrete	Concrete
Layer2	Cellulose fiber	Cement mortar	Cement mortar
Layer3	Cement mortar	Polyurethane exp.	Stiferite gt
Layer4	XLAM	Tuff	XLAM
Layer5	Cement mortar	XLAM	Plaster
U (W/m <sup>2</sup> K)	0.28	0.15	0.15
Y <sub>12</sub> (W/m <sup>2</sup> K)	0.0116	0.0066	0.0068
Y <sub>23</sub> (W/m <sup>2</sup> K)	5.32	2.68	3.31
Y <sub>34</sub> (W/m <sup>2</sup> K)	5.86	6.01	6.01
fd	0.041	0.041	0.045
M <sub>i</sub> (Kg/m <sup>2</sup> )	397.34	424.17	377.71
Δt (h)	17.95	16.26	17.81
κ <sub>1</sub> (J/m <sup>2</sup> K)	80.56	82.7	82.62
κ <sub>2</sub> (J/m <sup>2</sup> K)	73.02	36.9	45.54
d (m)	0.392	0.41	0.427
% Eco-sustainable	88.8	88.8	88.8

Table 2 - Heavy external walls - examples of high performance walls

WALL	W7	W8	W9
Layer1	Concrete	Concrete	Concrete
Layer2	Wood and hemp fiber panels.13	Polyester fiber.8	Hemp fiber
Layer3	Polyurethane exp.	Polyurethane exp.	Wood and hemp fiber panels.12
Layer4	Wood and hemp fiber panels.6	Wood and hemp fiber panels.6	Plaster
Layer5	Plaster	Plaster	Cement mortar
U (W/m <sup>2</sup> K)	0.12	0.10	0.10
Y <sub>12</sub> (W/m <sup>2</sup> K)	0.0128	0.0107	0.0089
Y <sub>23</sub> (W/m <sup>2</sup> K)	2.03	2.03	5.24
Y <sub>34</sub> (W/m <sup>2</sup> K)	5.84	5.82	4.46
fd	0.105	0.098	0.084
M <sub>i</sub> (Kg/m <sup>2</sup> )	229.1	228.3	208.5
Δt (h)	15.28	16.03	18.38
κ <sub>1</sub> (J/m <sup>2</sup> K)	80.3	80.0	61.3
κ <sub>2</sub> (J/m <sup>2</sup> K)	27.8	27.8	71.9
d (m)	0.347	0.375	0.42
% Eco-sustainable	88.8	88.8	88.8

Table 3 - Light external walls - examples of high performance walls



# Design and modeling the $\phi$ -F chart method for active solar energy systems in Scilab

Have you ever dreamed of keeping your home warm and cosy, getting hot water while saving money? Would you like to know how much solar energy you can exploit in your own heating system and carefully evaluate the financial cost?

The  $\phi$ -f chart method is a simulation technique used in solar energy systems for heating and cooling and can be used to compute the solar saving fraction for hot water production with respect to the global request. This model is completely parameterized and very fast; the advantage of having such a model is that it returns results within a few seconds, showing all the relevant system quantities necessary to calibrate an optimal system. It is also particularly useful for sizing systems components, whose main difficulty lies in the fact that these systems are influenced by both predictable data (e.g. collectors, storage tanks etc...) and unpredictable data, such as weather data. We developed an open source Scilab module for the study of closed loop solar energy systems (as reported in Figure 1) using the  $\phi$ -f chart method, that can be used to estimate the thermal performance of solar energy systems for domestic water, industrial processes heating, absorption refrigerators and space heating systems, where the thermodynamic cycle efficiency is independent of the heat supply temperature.

The system we are considering is composed by a solar collector array designed to collect heat by absorbing sunlight; subsequently, the heat is transferred to the storage tank through a heat exchanger which separates the fluid used in the solar collector from water contained in the storage tank and, eventually, the heat is transferred to the load through a circuit with an auxiliary heater, that can compensate to reach the minimum required temperature. The mathematical formulation for such

a nonlinear system involves 12 equations and 12 unknowns, which are solved in Scilab using the function "fsolve".

## Working example

An industrial process heat system located in Rome (41° 53' 35" N latitude) has a 100 m<sup>2</sup> collector tilted at 40° (south orientation) with the following characteristics:

$$F_R U_L = 4.5 \text{ W/m}^2\text{C}$$

$$F_R (\tau\alpha)_n = 0.85$$

$$F'_R / F_R = 0.95$$

where  $F_R$  is the collector overall heat removal efficiency factor,  $U_L$  is the overall energy loss coefficient,  $(\tau\alpha)_n$  is the transmittance-absorptance product for radiation at normal incidence and  $F'_R$  the corrected  $F_R$  factor. The process requires heat at a rate of 20kW at a temperature of 70°C for 8h each day. The specific storage is 120 kg m<sup>-2</sup> of water and the tank has an overall loss coefficient  $(UA)_s = 14 \text{ WK}^{-1}$ . The heat exchanger has

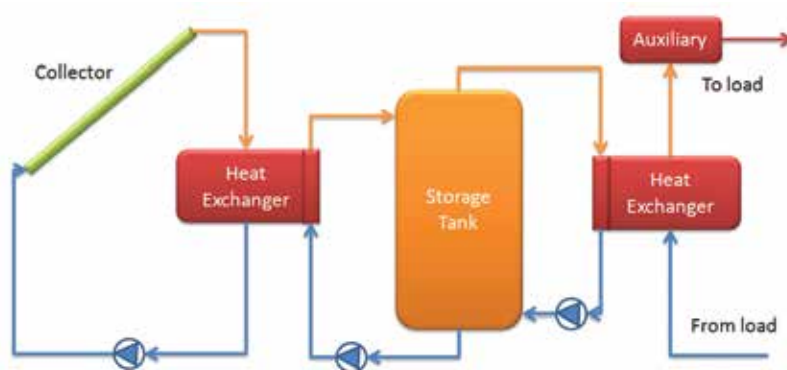


Fig. 1 - Schematic diagram of a closed loop solar energy system



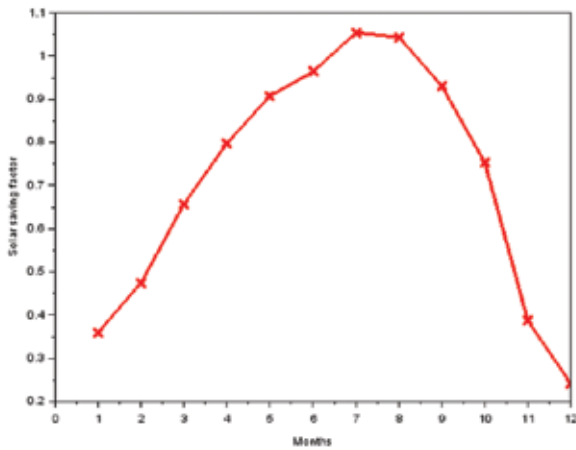


Fig. 2 - Solar saving fractions during the 12 months

effectiveness  $\epsilon_L=0.75$  and minimum capacitance of the two fluid  $C_{min}=5000$  W/K.

Once we have set all data related to solar radiations and temperatures in Rome during the 12 months, we run the simulator and visualize the results. For instance, Figure 2 shows the solar saving fractions during the 12 months.

We can notice that in July and August the solar saving fractions reach values higher than 1, which means that during these months solar energy satisfies the total energy demand.

### Varying the parameters

The previous example is here simulated by varying one parameter at a time. The following figures report the solar fraction for hot water production during the year.

Figure 3 shows the solar fraction during the year for three different collecting areas: 80m<sup>2</sup>, 90m<sup>2</sup>, 100m<sup>2</sup>. It is evident that increasing the collecting area, the fraction increases. Moreover, we notice that the highest fraction in Rome is reached in July. This is due to the maximum value of the incident solar radiation.

Figure 4 describes the behavior of the solar fraction during the year for three different tilt angles of the solar collector: 20°, 30° and 40°. It is visible that configurations with lower angles are better during the summer and higher values of angles are better during the winter. This is due to the sun's apparent position during the year. This simply means that a fixed tilt angle cannot be optimal for all the months and that an automatic change in the inclination could be advisable.

Figure 5 shows the behavior of the solar fraction during the year for three different specific storages: 80 kg/m<sup>2</sup>, 100 kg/m<sup>2</sup>, 120 kg/m<sup>2</sup>. This behavior is very simple: the more the mass, the

more the fraction. The effect is more visible during the summer than during the winter.

Figure 6 contains the last set of experiments that have been carried out by changing the minimum required temperature. The experiments have been done for three different values: 60°C, 65°C and 70°C. The effect is simple all year round: the less the requirement, the higher the solar fraction.

### Conclusion

We developed a Scilab module for the study of solar energy systems using the well-known  $\phi$ -f chart method. This model is completely parameterized and very fast. The advantage of having such a kind of model is that it may return within few seconds all relevant system quantities necessary to calibrate an optimal system.

The presented model is intended to be integrated with costs, allowing engineers to design the most appropriate solar energy systems at the minimum cost. This can be done directly within Scilab using one of the available optimization algorithms.

Anna Bassi, Manolo Venturin - EnginSoft

Ask the expert in Scilab:  
Anna Bassi, EnginSoft  
a.bassi@enginsoft.it

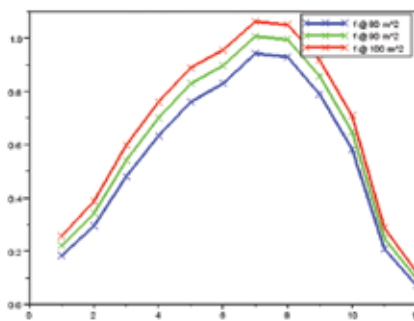


Fig. 3 - Solar fraction with different collecting areas solar collector

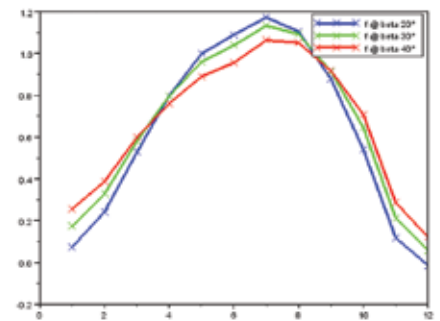


Fig. 4 - Solar fraction with different tilt angles of solar collector

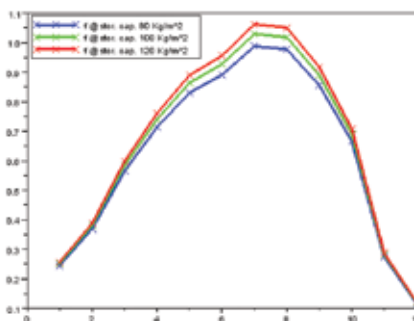


Fig. 5 - Solar fraction with different specific storages

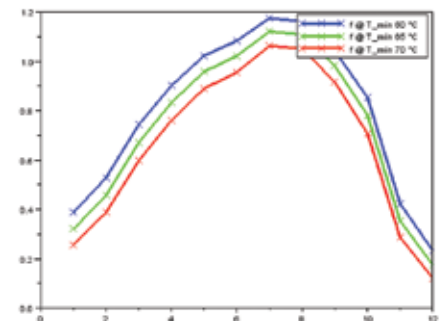


Fig. 6 - Solar fraction with different minimum temperature requirements

# Simulation governance

## An idea whose time has come

There are substantial economic incentives to reduce reliance of physical testing and increase reliance on numerical simulation. The reasons for this are obvious: Testing is expensive and time consuming and the results of tests are tied to the specific conditions under which they were performed. Testing without a properly formulated plan about how the results will be interpreted and generalized makes no sense. The main points are discussed in the following.

### Simulation

Simulation is the imitative representation of the functioning of one system or process by means of the functioning of another. Here we consider the functioning of mechanical and structural systems and their imitative representation by mathematical models. We view simulation as a transformation of one set of data  $D$  into another set  $F$  using the available background information and know-how  $I$ . In short hand:  $(D, I) \rightarrow F$

An important question that professional analysts must be able to answer is this: Are  $D$  and  $F$  such that they lie within the range of validity of the assumptions incorporated in a mathematical model? For example, in the classical beam model shear deformation is neglected. For what range of  $D$  is this permissible? How can we tell whether a particular  $D$  lies within range of validity of the model? What to do when  $D$  is outside of the range of validity of the model?

### Numerical simulation

The purpose of simulation is to predict  $F$ . However, owing to the complexity of the vast majority of practical problems, the best we can do is to find an approximation to  $F$  by numerical means, usually by the finite element method. We denote this approximation by  $F_{num}$ . The main elements of numerical simulation are illustrated in Figure 1.

Numerical simulation raises the obvious question: Under what conditions are we justified in substituting  $F_{num}$  for  $F$ ?

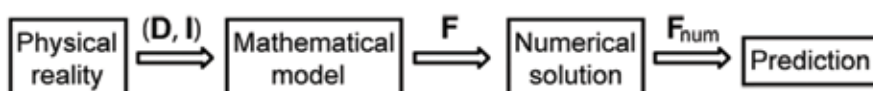


Fig. 1 - The main elements of numerical simulation

Let us assume that we have formulated a mathematical model and it is known that  $F$  exists and is unique. ( $F$  exists only if the data of interest are finite numbers). Note that  $F$  exists independently from the numerical method used for finding  $F_{num}$ . Furthermore, the numerical method must have the property that, as the number of degrees of freedom  $N$  is increased, the difference  $F - F_{num}$  decreases. In the finite element method  $N$  is typically increased by mesh refinement, increase of the polynomial degree of the elements, or both.

$F_{num}$  cannot be close to  $F$  if it changes substantially with  $N$ . This is necessary condition for the acceptance of  $F$ . Reporting  $F_{num}$  without (a) knowing that  $F$  is a finite number and (b)  $F_{num}$  is substantially independent of  $N$  should never be acceptable.

### Finite element modeling is not numerical simulation

The term finite element modeling refers to the practice of selecting elements from the finite element library of an FEA software product and constructing from those elements a numerical problem that is supposed to represent some physical reality.

A schematic representation of finite element modeling is shown in Figure 2 where the question mark indicates that it is generally not clear what the underlying mathematical model is.

In our short hand notation:  $(D, I) \rightarrow F_{num}$

Many practitioners of finite element modeling mistakenly believe that if two nodes of any combination of elements in a finite element library can be connected then the resulting assembly of elements will correspond to an approximation to a well-defined mathematical model.

The following example of a finite element model intended for simulating the mechanical response of an adhesive in a bonded joint illustrates this practice:

*"The method uses a gap the thickness of the adhesive, two rigid elements, and three zero-length spring elements between coincident nodes. One rigid element stretches from one adherend to a node at the center of the gap, while the second rigid element stretches from the other adherend to a coincident node also at the center of the gap..."*

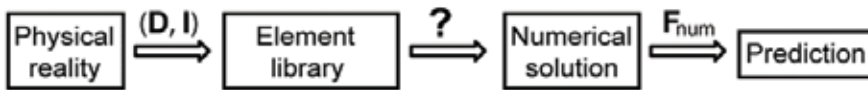


Fig. 2 - The scheme of "finite element modeling"

What is wrong with this approach? – It is not clear how rigid elements and zero length spring elements represent the mechanical response of an adhesive. The mathematical model is not defined, therefore it is not possible to estimate the difference  $F - F_{num}$ . In fact, if  $F$  includes the normal and shear stresses on the interfaces between the adhesive and the adherends then  $F$  does not exist. Therefore  $F_{num}$  will be an approximation to a non-existent  $F$ .

In the points where rigid elements are connected to an adherend the strain would be infinitely large in a mathematical model based on the theory of continuum mechanics. However a finite element solution will produce finite strains which would change as the mesh is refined. In other words, the finite element solution would strongly depend on the choice of the mesh. Two analysts using different meshes would be more than likely to come up with very different answers.

Practitioners of finite element modeling are often perplexed by "non-physical hot spots" observed at point constraints and at re-entrant corners. These are manifestations of conceptual errors associated with finite element modeling.

### Correlation with experiments

Suppose that an experiment is to be performed to test the veracity of predictions based on a mathematical model. We denote the measured data by  $F_{exp}$ . We are interested in the difference  $F - F_{exp}$ . However we have available  $F_{num}$  only. By adding and subtracting  $F_{num}$  we have:

$$F - F_{exp} = (F_{num} - F_{exp}) + (F - F_{num})$$

This means that we are justified in relying on the first bracketed expression on the right if and only if it is shown by independent means that  $(F - F_{num})$  is smaller than the errors in the experimental measurements. This is a key technical requirement, called solution verification, which is a prerequisite for model validation.

Since  $F$  does not appear in finite element modeling, and, in fact it may not even exist, the predictive performance of finite element models cannot be tested.

Numerical simulation is concerned with the formulation of mathematical models and the corresponding data of interest  $F$  which are approximated by numerical means. Finite element modeling, on the other hand, fails to distinguish between  $F$  and  $F_{num}$  and hence does not meet the technical requirements of verification and validation. *That is why finite element modeling should never be confused with numerical simulation.*

### Simulation Governance

Simulation governance is the exercise of command and control over all aspects of numerical simulation. Reliance on predictions based on mathematical models is justified if and only if convincing experimental evidence is available demonstrating that the transformation  $(D, I) \rightarrow F$

yields predictions that are consistently confirmed by the outcome of physical experiments.

Simulation governance is concerned with the selection and adoption of the best available simulation technology, the formulation of mathematical models, the management of experimental data, the verification procedures,

and the revision of mathematical models in the light of new information collected from physical experiments and field observations.

The key elements of simulation governance are Verification, Validation and Uncertainty Quantification (VUQU).

Verification is concerned with the accuracy of the numerical solution of a mathematical model and encompasses code verification, data verification and solution verification. Whereas code verification is the responsibility of the code developer, data verification and solution verification are among the most important responsibilities of the users. Software products qualified to support simulation governance must provide the means users need to exercise those responsibilities.

Validation is concerned with objective assessment of the predictive performance of mathematical models by comparing simulation results with the outcome of experiments, while accounting for the effects that uncertainties in  $(D, I)$  have on the data of interest  $F$ .

Uncertainty quantification is concerned with the stochastic aspects of  $D$ . In solid mechanics mathematical models are typically deterministic but the input data are stochastic. Therefore predictions based on mathematical models have to be stochastic.

### Closing remarks

With very few exceptions, physical testing is being conducted without a properly formulated plan for the generalization of the test results. Consequently the return of investment on testing programs is much smaller than it would be had testing been conducted under the rules of simulation governance.

For example, a great deal has been invested into the development of design rules for composite materials, yet the predictive performance of mathematical models conceived to support design and certification of structural components made of composite materials has not been satisfactory.

The available data  $D$  and background information  $I$  change over time. Therefore it is possible to improve the predictive performance of mathematical models over time. It is the responsibility of engineering management to establish a framework for simulation governance that includes systematic review and revision of simulation practices in the light of new information. Finding a nearly optimal balance between physical testing and numerical simulation is a problem of resource allocation. This problem can be solved only if the software tools used for generalization of the test results meet the technical requirements outlined herein.

Barna Szabó, Ph.D., PE President, ESRD, Inc.

For more information:  
 Marco Perillo, EnginSoft  
[m.perillo@enginsoft.it](mailto:m.perillo@enginsoft.it)





# Easy Geometry Preparation in ANSYS SpaceClaim Direct Modeler

## Why SpaceClaim?

Today, more than ever, company resources are being stretched further by business demands, resulting in fewer people available to handle large workloads, in tighter schedules and smaller budgets. Product development has to be carried out in ever shorter time cycles, requiring the use of quick and easy-to-learn software to aid the development process.

ANSYS SpaceClaim Direct Modeler (SCDM) is a useful 3-D solid modeling tool, suitable for analysts who handle complex 3-D models, but are not experts in traditional feature-based CAD systems. It enables product development and design engineers to modify 3-D geometry models. As a result, simulation can be conducted earlier in the product development process, where it can have the most impact on performance and cost.

## What is Direct Modeling?

Constraints embedded in historical geometries can complicate the modeling process, making model simplification and preparation time-consuming. SpaceClaim works around this issue with its streamlined, highly easy-to-use GUI, without the constraints and intricacies of historical geometry modeling. Direct modeling revolutionizes the way engineers think about working with 3-D solid models, enabling them to focus on designing rather than working around the constraints of their parametric CAD tool. Unlike traditional, feature-based CAD, ANSYS SpaceClaim Direct Modeler software allows users to reuse data and easily incorporate geometry from other designs, even those created in CAD



systems. Historical models often fail to regenerate if the original designer did not anticipate making the change in question. This is a severe limitation for analysts working with geometry models passed “over the wall” from the CAD team.

## SpaceClaim in ANSYS Workbench

ANSYS SpaceClaim Direct Modeler works both in stand-alone mode and as a Geometry cell in an ANSYS Workbench project. SpaceClaim is the geometry tool complementary to DesignModeler

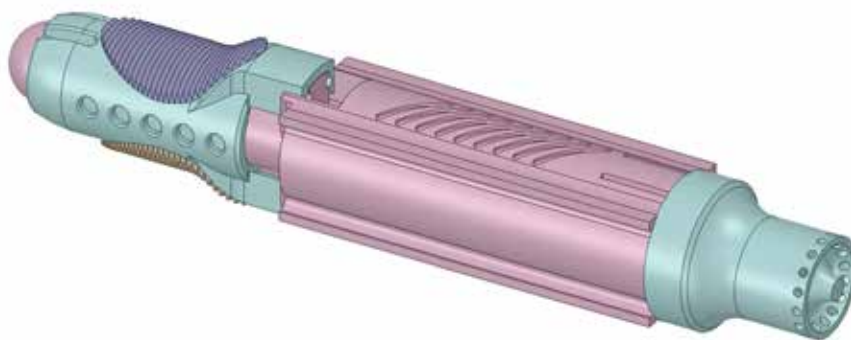


Fig. 1 – ANSYS SpaceClaim Direct Modeler software interoperates with all major native CAD and neutral file formats

which can be embedded in a Workbench CAE project. If DesignModeler is a history-based parametric application, SpaceClaim is a direct modeling application. Thus users can now choose how to handle their geometry. If DesignModeler excels at detailed parametric design and making global changes to a model, SpaceClaim excels at ease-of-use and making local changes to an existing model, aside from handling complex geometries.

### Geometry Import

The source of the starting geometry is not an issue, since analysts can work with any geometry, no matter where it comes from. Indeed, SpaceClaim is able to import geometry from all the main commercial CAD products. The list of importable file types follows: ACIS, AMF, AutoCAD, CATIA, DesignSpark, ECAD, IGES, Inventor, NX, OBJ, Parasolid, Pro/ENGINEER, Rhino, SketchUp, Solid Edge, SolidWorks, STEP, STL and VDA. Furthermore, the prepared model can be saved with file extensions other than the common CAD formats, such as AutoCAD files, Rhino files, STL, VRML. The advanced import-export capabilities described make the tool an efficient CAD translator, unlike many alternative products on the market.

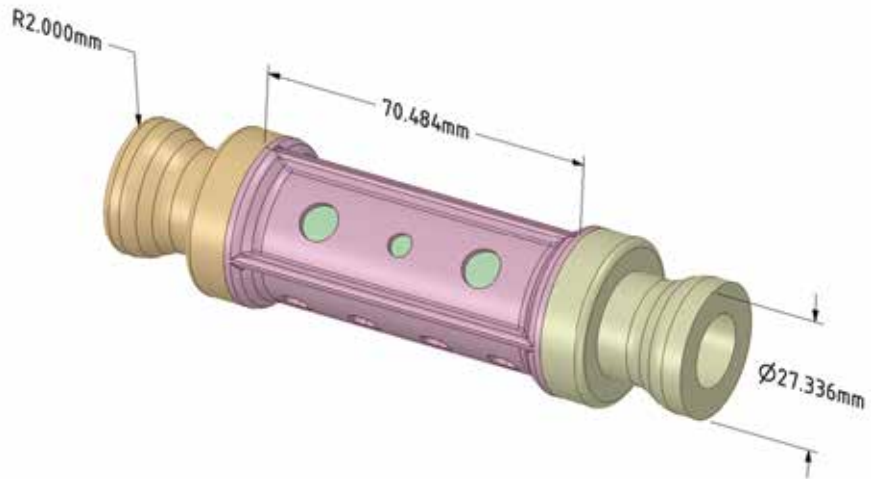


Fig. 2 – Almost any dimension in the geometry can be parametrized and the driving dimensions are automatically recognized in ANSYS Workbench

### No-Constraints Parametric Models

The geometrical model can undergo variations in a parametric workflow. Almost any dimension can be upgraded as a driving parameter of the model and remain a guiding element while the geometry goes through modifications. Moreover, due to the advantage of direct modeling, there is no need to set up complicated systems of constraints to handle a parametric model. Any driving dimension inserted in the geometry is automatically recognized by ANSYS Workbench, and is treated as all other parameters in the unique interface which allows full project parameterization.

### CAE Model Preparation

SpaceClaim provides CAE analysts with an easy and fast tool to get geometry ready for simulation. Many de-featuring operations are present, in order to aid model clean-up. Many details are often unnecessary to achieve accurate results from simulations, and SpaceClaim has robust simplification tools for the removal of rounds, imprints, depressions, protrusions and interference regions. Furthermore it is possible to repair surface bodies, split faces, inexact edges, duplicate entities and many more. The easy-to-learn workflow for geometry clean-up allows analysts to save days or weeks of work by quickly removing CAD bottlenecks in the CAE model. For the CFD analyst, the fluid domain can be extracted using the Volume Extract tool and the Enclosure tool, specific for internal and external fluid dynamics models. Once the geometry is clean, the bodies can be grouped together sharing their topology, in order to guarantee continuous mesh in separate domains, facilitating the following analysis. Finally, when the de-featuring operations are completed and the assemblies are correctly specified, the SpaceClaim model is read in the ANSYS Workbench environment and can be easily transferred to a mesher for grid generation.

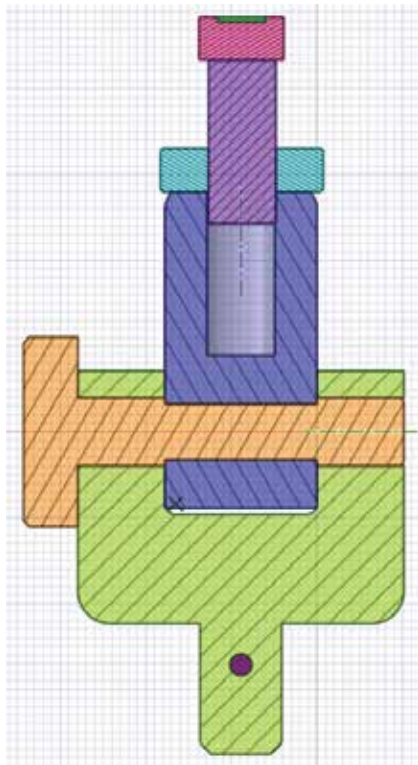


Fig. 3 – Among the software capabilities in handling large models, the possibility of investigating cross sections and modifying the geometry in such design mode

Ask the SpaceClaim expert:  
 Sebastian Colleoni, EnginSoft  
[s.colleoni@enginsoft.it](mailto:s.colleoni@enginsoft.it)



# Simulation of combustion system

## Reaction Design's Products

From 2015 onwards EnginSoft will be distributing and supporting the Reaction Design's suite, recently acquired by ANSYS. Understanding and predicting the effects of chemistry in a combustion system is key to developing competitive products in transportation, energy and materials processing applications.

Reaction Design technology helps design engineers to achieve their performance and efficiency goals with comprehensive software simulation tools. CHEMKIN's suite can be used for standalone simulations in order to develop detailed mechanisms, to predict the chemical performance, to create fuel surrogate formulation and to automatically reduce reaction mechanisms in several industrial sectors. FORTÉ CFD Package delivers accurate simulation of combustion in IC engines. ENERGICO software brings accurate chemistry simulation to combustion systems using automated reactor network analysis. Reaction Design also offers CHEMKIN-CFD to extend the power of CHEMKIN to CFD and enables the introduction of more accurate chemistry into reacting, fluid-flow simulations performed with ANSYS' FLUENT.

The User Interface allows the user to set-up the problem by visualizing the reactor-network diagrams.. The built-in visualization options provides quick graphic representation of the results, as well as the ability to easily export data for use in 3rd-party analysis tools, such as Excel. CHEMKIN PRO's advanced solvers and full feature set support the rapid and accurate development of models for specific applications, specifically designed for chemical kinetics simulations requiring complex reaction mechanisms. For the simulation of less complex systems, CHEMKIN is a robust, mature chemistry.



Detailed chemical models of combustion involve elementary reactions, where reaction paths represent molecular interactions and the breaking and forming of chemical bonds. Such chemical models provide accuracy and extensibility in predicting trace species emissions, critical chemical phenomena such as ignition or catalytic "light-off", and materials deposition and chemical modification. CHEMKIN is used both to develop detailed mechanisms and to predict the performance of the chemistry in pre-defined flow conditions with a large choice of Reactor Models.



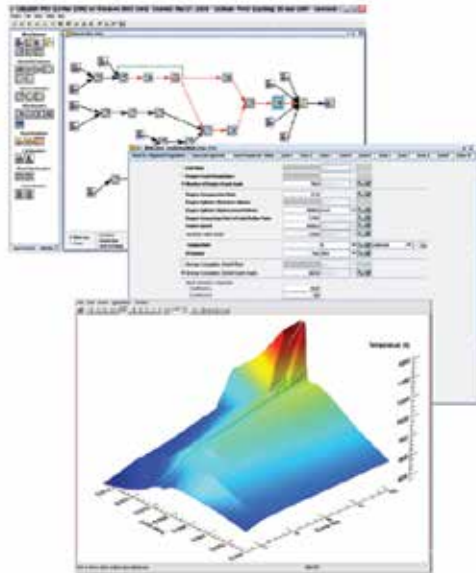




## Reaction Workbench

Reaction Workbench is an extension to CHEMKIN-PRO that enables automatic mechanism reduction.

Using the right fuel chemistry model for the job is critical in getting accurate answers within a project's time constraint. Master mechanisms can be automatically blended from component mechanisms, or sub-mechanisms can be extracted, using Reaction Workbench's Mechanism Utilities. For mechanism reduction, Reaction Workbench allows full user control over the automated process. Users select the specific simulation targets of interest (e.g., flame speed, ignition delay, NO<sub>x</sub>, soot, etc.) and the maximum allowable error for each target. Reaction Workbench automates application of a series of mechanism reduction techniques to meet those simulation objectives. The result is a mechanism that is optimal for the job and a direct measure of the reduced mechanism's accuracy relative to the original and can be used in any CHEMKIN-compatible product such as the FORTÉ CFD Package, the ENERGICO simulation package or CHEMKIN-CFD package. Reaction Workbench includes a Surrogate Blend Optimizer utility that allows the specification of the proper blend of pure components that match specific physical and chemical properties.



ENERGICO software translates the results of a CFD simulation into an Equivalent Reactor Networks (ERN) that captures the dominant flow patterns and recirculation zones. The use of ERN allows the flow field of the combustor to be represented as a group of ideal reactors with associated connecting streams so detailed combustion chemistry can be applied in an efficient manner to simulate trace species formation and destruction.

The high degree of accuracy afforded by ENERGICO's approach can be used to solve the toughest combustor and burner engineering challenges related to emissions reduction and stability even the possibility to evaluate the impacts

of variations in operating conditions, such as fuel composition, on combustion performance.

ENERGICO software is compatible with popular commercial CFD codes such as FLUENT and CFX. The ENERGICO package only requires a standard CGNS file as input, so creating a link from virtually any CFD code is easy.



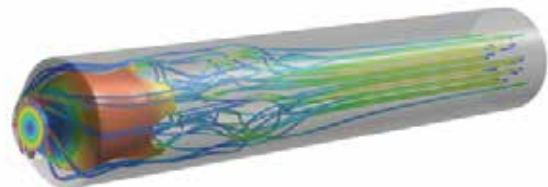
## ENERGICO

Combustor design simulation requires high resolution of complex geometry, turbulent flow patterns, heat transfer and detailed chemistry. CFD is the dominant tool for combustion simulation that supports the design of combustors and burners. However, CFD often falls short in the prediction of critical performance metrics such as CO and soot emissions, and Lean Blow Off (LBO) tendency. The inaccuracies in CFD's prediction of key phenomena can be directly linked to the limitation of most CFD software on the degree of detail that can be included in fuel models. In most cases, the models must be severely reduced in order to achieve a reasonable Time-to-Solution. The reduced chemistry typically used in CFD cannot represent trace pollutant species such as NO<sub>x</sub>, CO and Unburned Hydrocarbons (UHC). To resolve these species requires using detailed reaction mechanisms with hundreds of species and thousands of reactions.



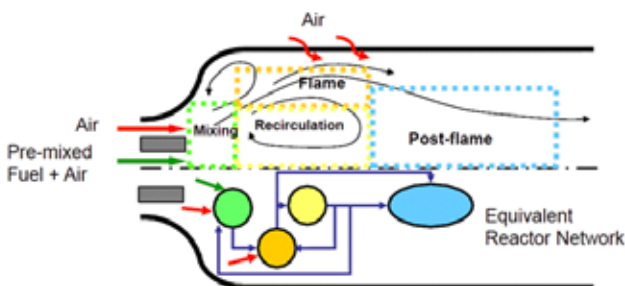
## CHEMKIN-CFD

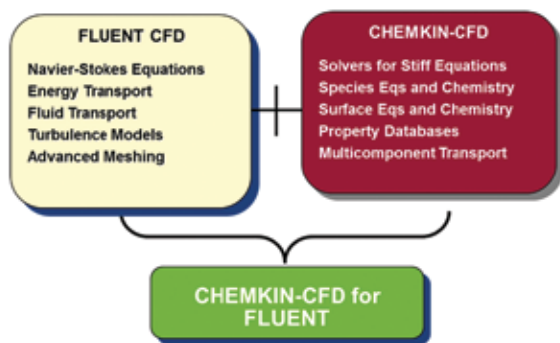
CHEMKIN-CFD is a plug-in chemistry solver that can be linked to other computational software packages, such as ANSYS' FLUENT CFD software, to add accuracy, speed and stability to calculations using finite-rate, multi-step reaction kinetics.



Detailed reaction kinetics are frequently required to represent combustion and materials process reactions accurately. Such accuracy typically means using multi-step reaction kinetics. However, using detailed kinetics often causes problems because of computational stiffness, where small changes in one variable may cause large changes in another. In the past, stiff chemistry simulations were characterized by long computational times and unstable solutions that often did not converge.

ANSYS, Inc. and Reaction Design have partnered to provide an advanced capability for simulating detailed chemistry in multidimensional, chemically reacting flows. The resulting technology closely couples Reaction Design's CHEMKIN-CFD chemistry solver with flow simulation capabilities in ANSYS FLUENT software, facilitating highly accurate combustion and emissions analysis for automotive engine and industrial power design. The analysis can be steady state or transient, and two dimensional or three dimensional.



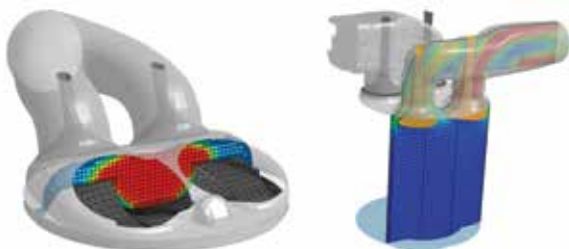


The CHEMKIN-CFD module for ANSYS FLUENT technology comes with several sample multi-step reaction mechanisms that are useful as a starting point for many combustion applications. The CHEMKIN-CFD module, licensed by Reaction Design, is available by request to ANSYS FLUENT licensees for free and will be included with each installation of FLUENT version 6.2 or higher.



**FORTÉ**

Today's engine designers face challenges in the design of low-emissions, high-performance vehicles that can run on a wide variety of fuels. New engine design strategies for emissions control and performance improvement, make fuel and pollutant chemistry critical factors that can determine overall engine performance. Combustion CFD simulation must be able to account for real fuel effects using the appropriate amount of kinetics detail in order to be a trusted part of the engine design flow.



FORTÉ robustly and accurately simulates IC engine combustion performance with nearly any fuel, helping engineers rapidly design cleaner burning, high-efficiency, fuel-flexible engines. Supported Engine Simulation applications include:

- Spark Ignition – Natural Gas, Gasoline Direct Injection & Port Injection.
- Diesel – Heavy Duty, Passenger, Power Generation and Marine.
- Dual-Fuel.
- Four-Stroke and Two-Stroke.
- Homogeneous Charge Compression Ignition (HCCI).
- Premixed and Partially Premixed Charge.

Ask the expert in simulation of combustion system:  
 Claudia Quadri, EnginSoft  
[c.quadri@enginsoft.it](mailto:c.quadri@enginsoft.it)

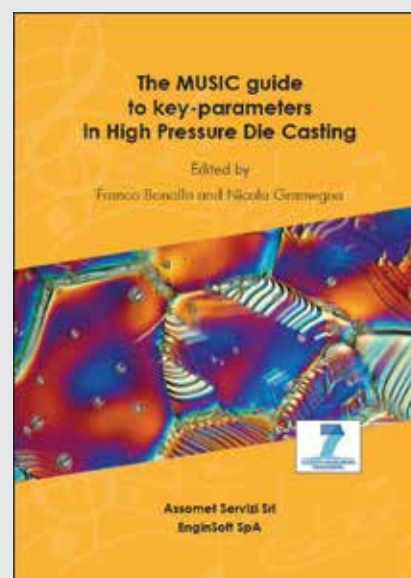
## MUSIC Public Training on Sensoring, Process Data Management and Process Simulation

### An innovative network to exploit the High Pressure Die Casting (HPDC) potential

The public training organized by the consortium of the MUSIC Project in the frame of the International CAE Conference was the first event open to the public in which the project could present its major achievements and results up to now. It was meant to share with an interested audience of possible stakeholders the project progress and its peculiarities, thus obtaining a great success with the participation of over 60 people, 35 of them external to the project, mainly manufacturing experts and end-users, daily involved in HPDC processes, looking for innovative solutions to control the injection process and to improve products quality, by optimizing resources and increasing production efficiency.

The public training course has also provided the opportunity and the right context to release the official first publication collecting the MUSIC results of the research obtained up to now: "The MUSIC guide to key-parameters in High Pressure Die Casting". This book collects the guideline to design and implement the Intelligent Sensor Network (ISN) in HPDC production line.

The success of the event is the demonstration of the great interest that this topic arises and it could be possibly repeated and enriched in another appointment out of Italy for European companies dealing in the manufacturing sector.



If you are interested in receiving your free copy, please send your request to [m.zanotti@enginsoft.it](mailto:m.zanotti@enginsoft.it) providing you contact details. For further information concerning the project: <http://music.eucoord.com/>



# The Intelligent Preventative Healthcare Solution

A sensor which raises the alarm if older people fall or have mobility problems is being developed by EnginSoft; who is working with Cranfield University on an initiative led by the Birmingham-based charity Housing & Care 21 to develop the Microsoft Kinect sensor for long-term preventative health and social care.

The Intelligent Preventative Healthcare Solution, which is supported by the Technology Strategy Board, is fitted in a lounge or a room where the most time is spent within their residence at the nursing home and captures the daily movements of the person that needs to be monitored. The software designed to help the UK's ageing population live independently for longer; can set off a trigger in the nursing staff office, if older people fall or if their mobility deteriorates. This means carers can provide the necessary assistance or if necessary call for medical help.

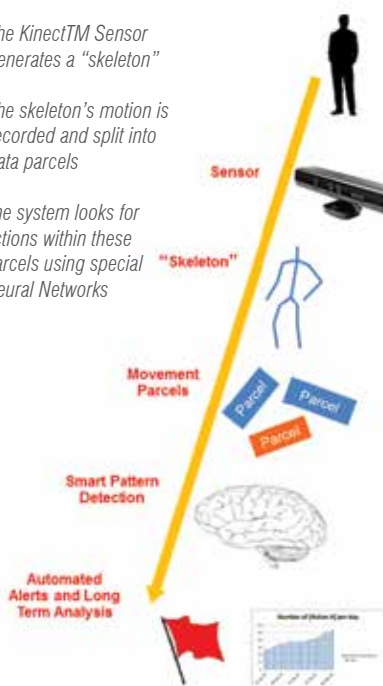
This solution considers the ageing population's determination to maintain their independence, who often do not want to be a burden or feel that they cannot look after themselves either at home or in a care home. The software provides a non-invasive ability to care for the elderly population whilst enabling them to remain independent.

The Microsoft Kinect sensor – the technology used by an Xbox – captures a 'skeleton' which represents the motion of each person's limbs to give a record of every day movement which means the data can identify significant incidents and long-term trends.

*The Kinect™ Sensor generates a "skeleton"*

*The skeleton's motion is recorded and split into data parcels*

*The system looks for actions within these parcels using special Neural Networks*



## The System In Use



The monitoring software identifies specific activities which it has been trained to detect so alerts are raised, for instance, falls or changes to their walking speeds over time which can signify a potential health problem.

Multiple sensors can be fitted but they are generally fitted in a living room since it can detect if people are moving around enough or if they are walking to other rooms. For example, if someone is not going to the kitchen for a drink on a regular basis, this can indicate that they may become dehydrated. The sensor will send data to where nursing staff are located who can schedule a visit and provide the appropriate care. The technology can be also be adjusted to spot for any areas of concern as people's needs change overtime.

Ask the expert in the healthcare project:  
Bipin Patel, EnginSoft  
[b.patel@enginsoft.com](mailto:b.patel@enginsoft.com)





# An overview of the FIRST project: CFD can be a valuable tool for a deeper understanding of the atomization process

## The challenge

Understanding the complex physics of spray atomization plays an important part in different industrial processes, from the design of low emission combustors to the cooling control for the rolling process in the metal industry. The characterization of an injector spray using today's CFD capabilities can be widely used to better the understanding of the design and development process.

## FIRST Project

The FIRST project (**Fuel Injector Research for Sustainable Transport**) was a collaborative project that ran for 48 months from December 2010 to 2014. Its 7.2M Euros budget was partially funded by the European Union 7th Framework Programme and 20 other industrial and academic project partners. The aim of the project was to deliver a technology that could reduce combustion emissions by developing an improved design tool/technique for modeling which could control fuel sprays and predict soot emissions in the aerospace sector. The project, coordinated by Rolls-Royce, involved 20 partners from 4 countries (UK, France, Germany, and Italy):

- 6 industries (Rolls-Royce UK, MTU, Rolls-Royce Deutschland, GE Avio, Snecma, Turbomeca)
- 4 SMEs (EnginSoft, ARTTIC, Scitek, A2 photonic sensors)
- 6 universities (University of Florence, University of Bergamo, Imperial College London, Loughborough University, KIT, University Pierre and Marie Curie)
- 4 research centres (ONERA, DLR, Cerfacs, CNRS)

Within aero-engine fuel injector design, spray distribution is a key factor in mixing fuel and air, which has a large impact on NO<sub>x</sub>, CO



and soot emissions. The FIRST project delivers a step change in the detail and accuracy of the spray numerical characterization, through new physical models and numerical methods. This enables soot reduction in a new generation of aero-engine combustors. First of all, the state-of-the-art experimental techniques for atomization and soot measurements have been used for the creation of a different and extensive database covering multiple aspects: fundamental configurations (to provide a new insight in the physical process of primary atomization), simplified injector configurations (to provide new empirical correlations) and



Fig. 1 - Partners in the FIRST project

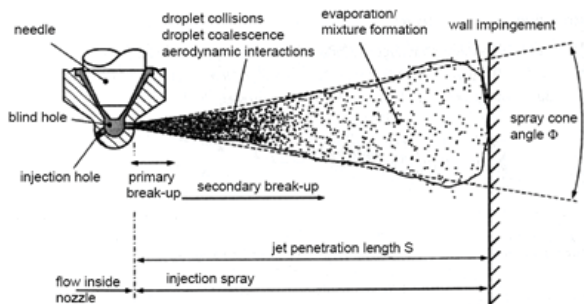


Fig. 2 - Spray break-up process

industrial injector configurations (to provide a population of fuel droplets according to different operating conditions and initial parameters). Then, development of physical models and numerical methods has provided reliable spray injection conditions for CFD codes in order to virtually reproduce the entire process as better as possible.

EnginSoft in the FIRST project has worked closely with GE Avio and the Universities of Florence and Bergamo for the development and optimization of break-up models and spray conditions. The expertise of the CFD team and the ANSYS CFX capabilities have allowed the usage of a commercial code in the field of sprays' atomization where usually research in-house dedicated codes are developed and used. The main topics investigated by EnginSoft are the entire break-up process including primary and secondary break-up atomization and the development of an expert system for the characterization of the spray boundary conditions in virtually every operating condition.

### Primary and Secondary Atomization

A correct prediction of the spray behavior requires a deeper understanding of the primary and secondary break-up processes. The main task of the primary break-up models is to determine the starting condition for the droplets that leave the injection nozzle. Primary atomization modeling has been an active research topic for the last two decades, where various numerical techniques for tracking interface motion and capturing the jump conditions at the interface have been studied. Within the FIRST project, different numerical techniques have been applied using realistic geometries, such as aeronautical injection systems, in order to evaluate the primary atomization. Before the FIRST project, the droplets were directly injected by fixed boundary conditions, whose characteristics were simply obtained by empirical correlations. EnginSoft, in collaboration with GE Avio, has tested the available models implemented in ANSYS CFX for the primary atomization and their behavior for the use with real injectors; in particular for high pressure swirl atomizers.

Moreover, in order to test the accuracy and reliability of secondary break-up and droplet coalescence/interaction models in ANSYS CFX, multiple operating conditions characterized by different values of chamber pressure and temperature and different flow rates for the injector, were investigated in the FIRST project by EnginSoft. The droplets behavior were tracked using the lagrangian method, including the liquid evaporation model and the turbulent dispersion force to account for the interaction of every single droplet with the turbulence flow field. Global and local criteria were then used to evaluate the process, where the global criteria refers to the evaluation of the spray penetration and the spray angle and the local criteria were used for the evaluation of the droplets diameter distribution inside the computational domain for the comparison with the experimental data in the probing points. The available models implemented in ANSYS CFX were deeply compared in order to identify the best compromise between accuracy and computational time.

### Expert System for Liquid Break-Up based on Artificial Neural Network

After the validation of the ANSYS CFD tools, a full virtual injector system was developed by EnginSoft with the aim to evaluate the spray characteristics for non-measured points inside the experimental test matrix. This type of detailed modeling allows us to understand the interaction between different phenomena in the complex process of primary and secondary atomization. The commercial code modeFRONTIER was then used to generate response surfaces. The Response Surface Methodology (RSM)

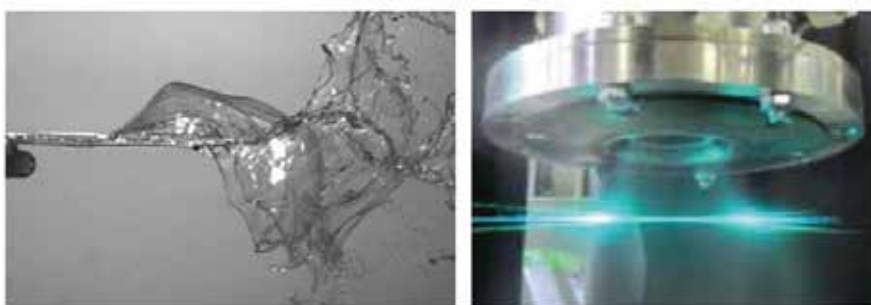


Fig. 3 - Example of primary breakup: Planar Liquid Sheet Break-Up (left) and High Pressure Swirl Atomizer (right)

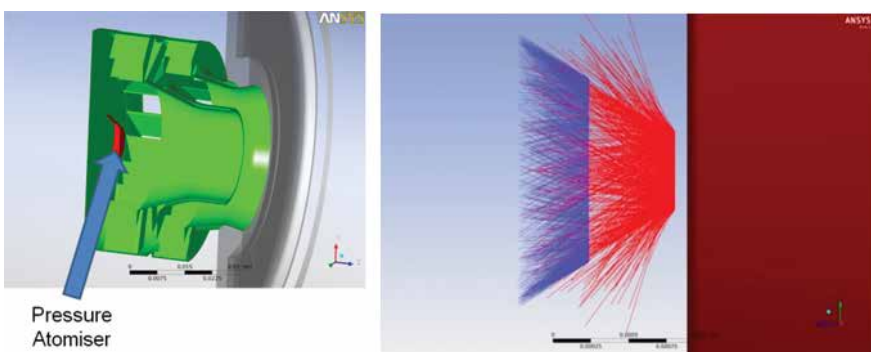


Fig. 4 - Comparison between ANSYS® CFX (blue) and academic research in-house code (red) for the evaluation of the primary atomization for an industrial injector configuration

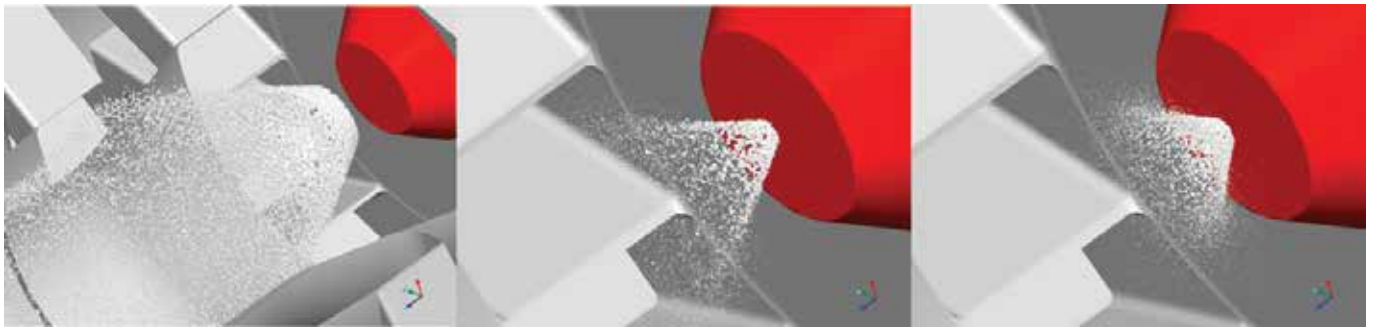


Fig. 5 - Break-up process for three ambient conditions: low pressure (left), middle pressure (center) and high pressure (right)

explores the relationships between input variables and one or more response variables and estimates the value elsewhere in the design space, on the basis of some assumption (regularity, physical meaning, and statistical variability). The input parameters (pressure and temperature of the chamber, flow rate of the injector) can be directly related with the characteristics of the spray (penetration, angle, Sauter Mean Diameter and Rosin Rammler coefficients) without performing additional accurate and CPU demanding CFD analysis. Only a reduced number of baseline conditions (design of experiment) are used to evaluate the response surfaces and then to identify the relationship between the parameters.

The method is suitable for the finding of new correlations that can be used from GE Avio in the development of future injectors and combustors. Traditionally the boundary conditions were based on empirical formulation and literature formulas, but now a more accurate evaluation tool is available with the RSM method. The correct prediction of the spray distribution is a key factor for the accuracy of the combustion process with the evaluation of NO<sub>x</sub>, CO and soot emissions and the Response Surface Method based on accurate CFD simulations can be a valuable tool.

**Acknowledgments**

The research leading to these results has received funding from the European Union 7th Framework Programme (FP7/2007-2013) under grant agreement n° 265848 (FIRST). For further details, please visit FIRST's website: <http://www.first-fp7project.eu>

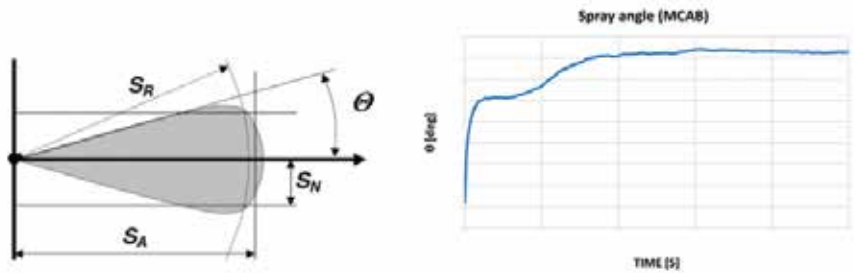


Fig. 6 - Example of Spray Angle (global criteria)

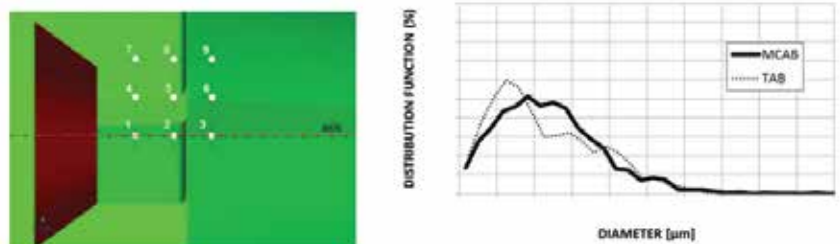


Fig. 7 - Probing points and example of droplets' distribution (local criteria)

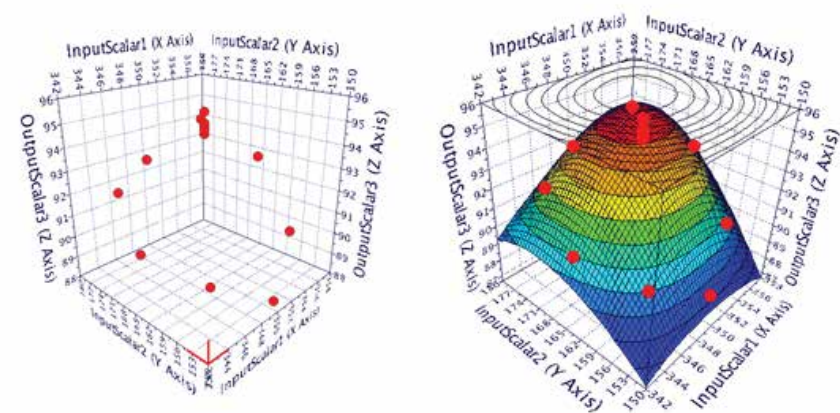


Fig. 8 - Response Surface Method: Design of Experiment (left) and Response Surface (right)

Ask the expert in First Project:  
Michele Andreoli, EnginSoft  
[m.andreoli@enginsoft.it](mailto:m.andreoli@enginsoft.it)





## The meaning of being partner

In 2014, CERATIZIT, world leader in the production of hard metal parts, requested EnginSoft's support in creating a Competence Centre for its customers in its Italian office located in Alserio (CO). The task was to train a group of technicians to support customers in CO-DESIGN activities by means of simulation. Our approach has been to first focus on software training (Forge\ColdForm of Transvalor), to be completed by an on-job program afterwards. Some specific cases suggested by customers were taken into account, thus creating a consciousness in the tool's usage. The first results of this activity were presented by Eng. Paolo Fusi of CERATIZIT in the "Metal Forming" session of the International CAE Conference: "Optimization of the mandrel profile for the production of rifled pipes and geometries modifications of a hard metal insert for nails cut".

CERATIZIT declares: "both analysis cases have allowed us to understand the origin of defects and to find feasible solutions".

Partnership means collaboration from both sides, and EnginSoft has accepted CERATIZIT's invitation to participate in a seminar at the office in Alserio (CO) on November 20<sup>th</sup> concerning the "Innovation in the cold metal forming processes". In this occasion the Competence Centre has been presented to many customers, giving evidence to the partners that have supported the initiative. Eng. Marcello Gabrielli of EnginSoft has presented the experience gained with CERATIZIT, the results obtained and other possible sectors in which the process simulation could be applied.

We thank CERATIZIT for this positive experience, it has been fruitful due to the collaboration we have had on our common interests. We will continue to tackle more challenges together.

*Marcello Gabrielli, EnginSoft*

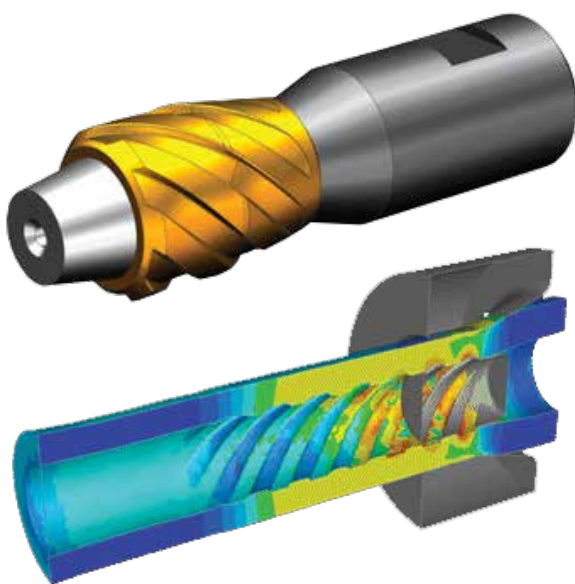


Fig. 1 - Mandrel used to produce rifled pipes and process simulation

Ask the expert in forging processes simulation:  
Marcello Garbielli, EnginSoft  
[m.gabrielli@enginsoft.it](mailto:m.gabrielli@enginsoft.it)



Fig. 2 - EnginSoft presentation in the CERATIZIT seminar



## TWO-DAY intensive workshop booked for ANSYS Finmeccanica users

A specialized workshop was held last November for ANSYS Finmeccanica users, which took place in Livorno at the Wass headquarters. Wass has been using the ANSYS suite of tools since 1987 and they are considered one of the leading companies to use ANSYS Workbench in Italy.

The event was driven by Pierre Thieffry (Product mechanical manager at the ANSYS headquarter) and Roberto Gonella (ANSYS mechanical product manager in EnginSoft – ANSYS Channel partner)

The aim was to share information about the current status of the ANSYS tool set and briefly discuss future ANSYS updates.

The recurring theme of the event was how can we make Workbench users into expert users? This can be summed up in the following way.

When comparing ANSYS 15 years ago to now, about 80% of its capabilities were being used back then, but now the utilization gap has widened even further. This is due to the fact that engineers prefer to use familiar procedures, even if in some cases they have been superseded by a better method. Nowadays Workbench has become a standard tool among ANSYS users who acknowledge that simulations are performed in an easier way, faster than before, while ensuring high accuracy of results. It is interesting to read the comments that some attendees have made about the event...

### **Ing. Schettino (CAE manager in OTO Melara)**

*“The ANSYS workshop for Finmeccanica’s users was held on the 18th & 19th November, attended by OTO Melara computation centre. It was an interesting opportunity to get an overview of Workbench’s functionalities and to exchange information with other companies who work in the same sector.*

*The presentation of the SpaceClaim Modeler and the upgrade of release WB16 have been the most interesting elements of the first day, whereas the lecture given by Marini about CFD codes given on the second day provided a focus on grid generation and boundary layer resolution.*

*The presentations concerning the grid generation in Workbench have underlined the advantages and disadvantages of the different types of grids and which tools support the quality control of the elements for a*



*well-configured computation.*

*The presenter also illustrated some of the basic and advanced features of the CFX and Fluent solvers. The FSI examples and the presentation of the different turbulence models implemented in the two solvers have been particularly appreciated.”*

### **Ing. Cardilli (ANSYS senior user at Mechanical Department in Selex)**

*“The CAE world has been changing in recent years, evolving from niche software tools to be used only by specialists into technological solutions which can meet the industrial world’s needs. In the case of ANSYS, the starting point has been the introduction of Workbench. Even if it was initially perceived as just a CAE tool integrated into CAD, and not a complete solution, it has quickly evolved by combining all calculation and simulation environments resulting in an actual multi-physic platform.*

*Leaving mathematic and physic technical details of the suite aside, in my perspective, it has been extremely important to dramatically reduce the time for analysts to reach the same design objectives obtained with previous technologies. This is probably the main opportunity that companies wish to take advantage of when using CAE as an integrated discipline in the design process.*

*In a company such as Selex ES, designing and producing electronic complex systems, there are several elements and functions that contribute to the idea and the structure of the devices, until the feasibility phase.*

*One of the main points that came out of the meeting in Livorno was*



that working with a CAE model from the very beginning of the product development and proceeding step by step with a well-structured process to get efficient and compliant results, is the real competitive advantage of this suite.

From a single product, ANSYS has become a suite with Workbench, which enables the specialist to follow the whole project development from the very beginning and linking all the physics elements in a unique environment, clearly and easily interconnected. The logical consequence of an integrated and efficient multi-physics suite is the great improvement provided as far as optimization is concerned. Nowadays it is possible, in the same amount of time, to perform more simulations in the workbench environment in comparison with the past. This has allowed the investigation of more solutions and to find the optimal, especially when it comes to reducing mass for a better product.”

**Wass: Giulia Filucchi (ANSYS senior user at Mechanical Department in Wass)**

“Last November, a group of ANSYS experts from EnginSoft and expert users from Finmeccanica met in WASS (Livorno) for a 2-day intensive Workshop. The first day was dedicated to ANSYS Workbench, especially to the Mechanical tools. It was an honour to meet the ANSYS manager for Europe, Pierre Thieffry, who showed the new capabilities of the ANSYS platform and tools in an interactive discussion. Together with the manager of the EnginSoft office in Florence, Roberto Gonella, Thieffry satisfied the curiosity of all the engineers involved in the Workshop, by answering

questions and addressing concerns. The second day was dedicated to an in-depth description of the powerful capabilities of the ANSYS CFD tools: which was music to the participants’ ears, who daily make use of ICEM, CFX, FLUENT and know well the “joys and sorrows” of studying complex CFD scenarios. The EnginSoft experts got to the heart of the matter and a great number of high-level technical issues were discussed. The presentation of the new Electromagnetic tools inside Workbench closed the Workshop. In addition to the technical content, the Workshop was for the Finmeccanica colleagues a valuable experience and an opportunity to exchange information, discuss common experiences, identify opportunities for cooperation and at last but no means least, maintain their working relationships into the future: that was the icing on the cake! So, many thanks to EnginSoft and WASS for organizing such an interesting event: professional growth activities are always welcome!”

..and eventually the written comment of Pierre ....democratization of code absolutely shines through his word...

**ANSYS Inc : Pierre Thieffry (ANSYS Mechanical product manager)**

“I like the level of interaction we had with the attendees of the workshop - most of them senior users with a strong understanding of the FEA technology. It is always useful for ANSYS to know what customers do with our products, as well as what they would like to do in the future or what they are missing. It is also good to be able to showcase advances in the product that users can quickly benefit from.”



**Training  
Center**

ANSYS MECHANICAL  
ANSYS CFD  
FLOWMASTER | ESACOMP  
modeFRONTIER | LS-DYNA  
MAGMASOFT | FORGE  
COLDFORM | FTI  
DYNAFORM | PLANETSX  
STRAUS7 | SCILAB

Drive your career!

2015

Training Program

[www.enginsoft.it/formazione](http://www.enginsoft.it/formazione)





# INTERNATIONAL CAE CONFERENCE

**1984 – 2014: thirty years ago, the FUTURE begins here ...**

**27th – 28th October 2014, Lazise (Hotel Parchi del Garda Congress Centre)**

The International CAE Conference, the major technical and scientific event in Europe and well-consolidated reference point for computer-aided simulation on a global level, has achieved the goals set out 30 years ago.

The 2014 conference has been the occasion to celebrate its 30th Anniversary. As usual it has been instrumental in disseminating knowledge about state of the art computational technologies, supporting research and technological innovation (for industry, aerospace, Oil&Gas, medical sciences), while allowing experts, researchers, technicians to discuss the latest issues in product and process innovation.

Almost a thousand people attended in the two day congress in Lazise, which consisted of a common plenary session, followed by 15 thematic parallel sessions. This was achieved thanks to the participation of 150 speakers coming from across Europe and around the World. They have shared with their audience the latest achievements of their research, the state of the art of simulation technologies, the future trends and the expected impacts in the industrial world. Forty companies have sponsored the conference activities.

This year the International CAE Conference was also supported by some eminent guests.

Confindustria President, Giorgio Squinzi, has declared: *“New computer-aided technologies have been acknowledged as essential for the creation of new processes and products and for this reason the European Commission is strongly focused on companies development. In such a scenario, companies have to play the fundamental role of key partners in research activities, thus stimulating innovation development and achieving Europe targets. Innovation is also an Italian priority and Confindustria is requesting the central government to foster the creation of favorable and suitable conditions for innovation and competitiveness”*. Stefano Odorizzi – CEO, Scientific Committee President of the

International CAE Conference – has underlined the importance of Computer Aided Engineering simulation and innovation as a strategic tool for competitiveness for any company in any sector.

Key-note speakers: the record man Francesco Moser, who gained the cycling hour record in Mexico City thirty years ago using an innovative bicycle for that time, equipped with lenticular wheels and adopting an elegant posture in the Pininfarina wind gallery; Matteo Trentin (Omega Pharma – QuickStep) the promising Italian cyclist.

They represented two generations of sportsmen that have allowed to rediscover the main innovations that have characterized the bicycle evolution and the simulation applied to cycling in general.

*Far away from being the gloomy year described by George Orwell in his famous novel “1984”, the world in 1984 proved to be rich in events that have deeply changed our present.*

*While Apple released its Macintosh, opening the way to the “everybody’s” computer, Mark Zuckerberg was born and Facebook would have caused a revolution in the worldwide connection; Televideo function introduced in Italy a brand new communication style; Francesco Moser gained the cycling hour record in Mexico City with 51,151Km while in the USA a baboon’s heart was for the first time transplanted on a baby, Baby Fae. In the same year Ronald Regan flew to Rome to present the project related to Alpha station (now International Space Station) and sign an agreement that allowed Italy to become the main European partner for that enterprise (over the 60% of the inhabiting volume of the orbiting station has been directly or indirectly produced by Italy).*

Each one of these events has been a seed for the “future”, as the International CAE Conference that, in 1984, was presented for the first time, and from that edition on it has delivered its contribution to science, research and knowledge industry, as well as to technological

innovation coming from computer-aided simulation, in a more and more relevant way.

Industrial production, energy, aerospace enterprises, medical sciences, catastrophe management: how much could our future and the quality of our daily life improve? Computer aided-engineering technologies represent both the answer and the way in which companies can move toward innovation and economic recovery for our World.

This edition has been characterized by the presence of eminent scientists, researchers, managers and engineers coming from all over the world and presenting the results achievements of some important research projects, as for instance:

**Professor Michael Gasik** - Aalto University, Helsinki (Finland) could demonstrate his scientific and biomedical theories during the International CAE Conference performing a live experiment. Gasik is renowned for his work finalized at exploiting the hydrogen available in the space, which represents the propellant for next generation of propulsion and will allow space exploration without constraints related to carrying fuel from Earth. **Kenneth Ruud**, University of Tromsø – Norway has focused his lecture on the recent discovery of huge deposits of natural resources located in the Northern Sea. Italy and Norway, thanks to the collaboration of their respective Ministries of Industry and Productive Activities, are finalizing techniques and innovative tools devoted to the exploitation of the resources available in the Arctic region with the minimum environmental impact possible.

**Carlo Sborchia**, division director of ITER project and distinguished Italian scientist, has presented the actual status of the ambitious European Project for energy production by nuclear fusion. The plant under construction in Cadarache – Southern France – is financed by the European Commission and involves scientists and engineers from all over the world.

This project is the European counterpart to an American one, similar in terms of investments and potential innovation in clean energy production, which aims at producing energy using solar light concentration by means of glasses and deflectors.

**Jérôme Lasvenes** - Airbus France – has dedicated his speech to the techniques currently in use on the modern A380, the biggest airplane ever built, able to transport 853 passengers, aimed at minimizing risks and guaranteeing maximum safety to passengers.



**Francesco Iorio** has talked about Expò 2015 and the avant-garde construction techniques, adopted to build the pavilions that will host next year's event.

As far as medical sciences is concerned, the session was opened by Davide Valtorta (Switzerland) presenting his work finalized at preventing heat diseases and in particular aortic valve deformations. Inmaculada Leyva – University of Madrid – has explained the results achieved by her research group, analyzing the complex structure of the brain neural system in its different morphologic and functional areas.

The International CAE Conference has always been acknowledged as a reference point for the academic, industrial and research world, as well as by the developers of simulation software solutions. This has been done ever since CAE has become an indispensable tool in all processes related to innovation, design and production in any field: medical sciences, aerospace, energy, transportation, material manufacturing, consumer goods, civil engineering and construction, biomechanics, high performance computing and many more.

The poster Award session has again supported the creativeness of young engineers: the prize specifically for university and PhD students has awarded the 5 best works (voted among the 40 finalists) selected from over 200 poster submissions to the Scientific Committee.

### Exhibiting Area

Every year, the exhibiting area has been the reference point for the main hardware producers in the global market. Just to mention some of them: IBM, Dell, Hewlett-Packard and specialists in High Performance Computing, not so famous to the wide audience, but well-known to sector experts. 3D printers were represented by application Engineers who were on hand to provide technical information and present the future design evolution. Lavazza was represented after its "Isspresso" collaboration with the International Space Agency: where a machine was developed to dispense coffee in the extreme conditions of space. The design had to cope with the absence of gravity and consider fluid-dynamics totally different from those experienced on Earth. Isspresso will be brought to the International Space Station by Italian astronaut Samantha Cristoforetti in November. The stand dedicated to cycling attracted a lot of visitors: Bicycles from the past, present and future were displayed, including the famous bike with lenticular wheels ridden by Francesco Moser in achieving his records. The lenticular wheel was introduced for the first time in 1984 by Francesco Moser in the hour record and then by the Italian Cycling National representative at the Los Angeles Olympic Games, winning the Gold medal in the 100Km. Electric and hybrid propulsion bicycles: The bicycle of Matteo Trentin, the promising Italian cyclist, used to win the last lap of the Tour de France.

Hi-Tech ultra-light components made of composite materials, light alloys and/or using innovative design and simulation techniques.

Last but not least, two automobile prototypes have been presented by the University of Padova and by the Politecnico of Torino, used during international race competitions among universities.

[www.caeconference.com](http://www.caeconference.com)



# International CAE Poster Award 2014: a showcase for young researches in love with innovation

## A prize dedicated to CAE technology application

The winners of the latest “International Poster Award: a poster for CAE” were awarded in Lazise, Garda Lake, on October 27th.

The prize, in its third year, is becoming a regular feature to promote numerical simulation amongst young engineers and grow the CAE analyst’s community. This is evidence of the great value that EnginSoft gives to the dissemination of scientific knowledge and providing talented young researchers with new opportunities.

The evening event, closing the first day of the International CAE Conference on “Simulation based Engineering and Sciences” was presented by Luca Viscardi, well-known Italian radio host, with the participation of two distinguished guests representing the virtual simulation world: Stefano Odorizzi, EnginSoft CEO and founder, and Bernhard Schrefler, emeritus professor of Construction Sciences at Padova University.

The different works were evaluated by a Scientific Committee, representing the most diversified sectors and academic research centres: Beatrice Belletti (University of Parma) Marco Evangelos Biancolini (University of Rome), Nicola Bonora (University of Cassino), Gabriele Angelo Dubini (Politecnico of Milan), Giorgio Fotia (CRS4 Bioinformatics Laboratory), Ugo Galvanetto (University of Padova), Michael Gasik (Aalto University), Gianluca Iaccarino (Stanford University), Giuseppina Montante (University of Bologna), Enrico Nobile (University of Trieste), Bernardo Schrefler (University of Padova) and Giorgio Zavarise (University of Salento). The pass-list was defined also considering the preferences given by the users who registered to participate in the on-line voting.

40 posters were selected among the many submitted projects and they were put up in a dedicated area inside the CAE conference and also published on-line in the Poster Award section on the Conference official web-site so to provide them with the widest visibility possible also once the event was over. The winning posters are presented in the following pages and their titles are:

- CFD Characterization of a Pediatric Ventricular Assist Device through a Moving Boundary Approach.
- Effect of volumetric flow rate on transition to turbulence in patient specific arteriovenous fistulae for hemodialysis.
- Enrichment of the Finite Element Method with the Meshless Local Petrov-Galerkin method in Structural Mechanics.
- HiMOD and HiPOD methods for solving direct and inverse problems in internal fluid dynamics.
- A fluid-structure approach to optimize the thrombogenic potential of artificial heart valves.

Prof. Odorizzi, active promoter of innovative ideas said to be very satisfied with the award success: *“I would like to underline the excellent level of the projects submitted in the last three editions and I would like to thank all the participants, since they have contributed to widening the use of CAE with their innovative ideas. I confess that it’s rewarding to know that some of these young researchers now have employment opportunities thanks to this initiative”*.

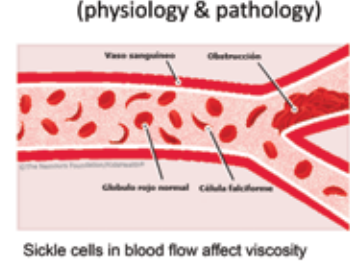
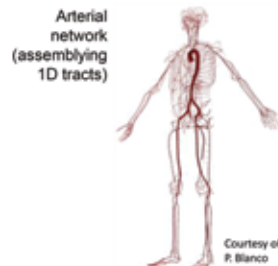
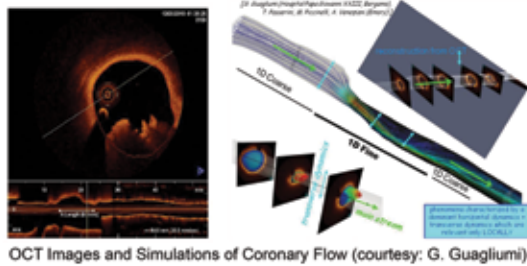
For further information go to [www.caeconference.com](http://www.caeconference.com), Poster Award section, or contact: [posteraward@enginsoft.it](mailto:posteraward@enginsoft.it)



# HiMOD and HiPOD methods for solving direct and inverse problems in internal fluid dynamics



Many practical problems involve incompressible fluids in pipes. For instance, in computational hemodynamics we need to



**WE NEED EFFECTIVE SOLVERS TO**

(1) LIGHTEN THE COMPUTATIONAL BURDEN (Direct Problems)

(2) SPEED UP CONSTRAINED MINIMIZATION (Inverse Problems)

DIRECT

## HiMOD = Hierarchical MODEL Reduction

$$u(x, y, z) = \sum_{i=0}^N U_i(x) \varphi(y, z)$$

FEM Geometrical versatility

Modal Analysis Low cost accuracy



### FACTS

- a) 1D FEM + Modal Convergence
- b) Adaptive selection of modes
- c) Extension to curved domains
- d) Convenient Algebraic Structure



### POD applied to Hi-Mod Reduced Problems aka HiPOD

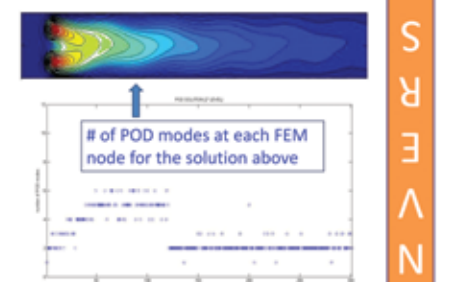
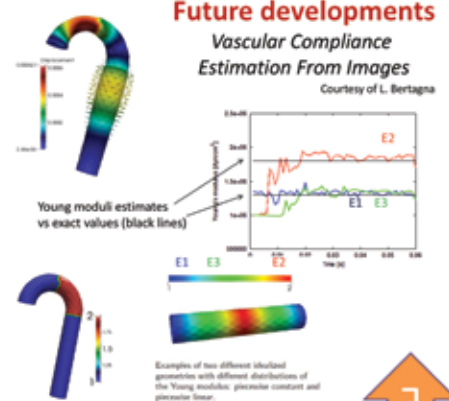
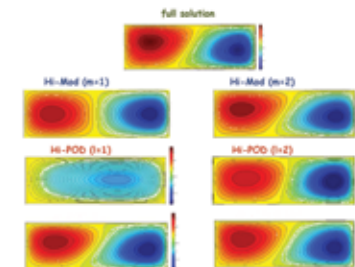
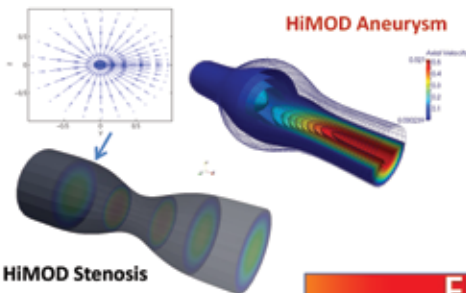
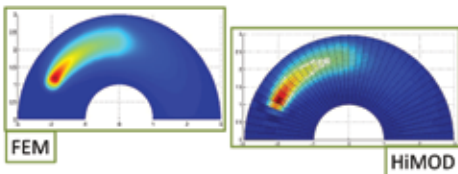
Aim: compute  $u_h(x, y, \alpha)$  without resorting to on line Hi-Mod procedures

Hi-Mod expansion

$$u_h^k(x, y, \alpha) = \sum_{i=1}^m \left[ \sum_{j=1}^{N_h} \hat{u}_{ij}^k \varphi_j(x) \right] \varphi_i(y)$$

$\hat{u}_{ij}^k = [\hat{u}_{1ij}^k, \dots, \hat{u}_{N_h ij}^k, \hat{u}_{i1}^k, \dots, \hat{u}_{iN_h}^k]^T \in \mathbb{R}^{mN_h}$

Prediction  $\rightarrow u_h^k(x, y, \alpha) = \sum_{i=1}^m \left[ \sum_{j=1}^{N_h} \hat{u}_{ij}^k \varphi_j(x) \right] \varphi_i(y)$



Proper Orthogonal Decomposition

Method for representing a parametric solution with a few degrees of freedom.

$$u = \sum_{i=1}^N U_i \varphi_i$$

Basis functions computed by

- $\rightarrow$  Offline preprocessing to create knowledge
- $\rightarrow$  SVD of the snapshots to reduce redundancy
- $\rightarrow$  Online step to estimate the parameter (inverse problem)

FROM DIRECT TO INVERSE

ESRANI

CAE POSTER AWARD 2014: WINNER

Bibliography: Perotto, Ern, Veneziani, SIAM MMS 2010; Perotto, Lect N Comp Sc Eng 2014; Perotto, Veneziani, J Sci Comp. 2014 Ackn: NSF Grant DMS-1419060 MOX, Politecnico di Milano, Italy & Emory University, Atlanta, GA USA - M. Aletti, A. Barone, S. Guzzetti, M. Lupo Pasini, S. Perotto (PI, Italy), A. Veneziani (PI, USA)

# CFD Characterization of a Pediatric Ventricular Assist Device through a Moving Boundary Approach



## Introduction

Cardiovascular diseases are recognized as the main cause of death in the world, primary due to heart failure [1]. Transplantation is still the gold standard to treat these pathologies; however, limited availability of fresh organs automatically requires the adoption of Ventricular Assist Devices (VADs) as a bridge to transplantation. In the pediatric field, appropriate CADs are currently under development in order to reduce thrombotic potential typical of rescaled adult devices [2].

## Aim of study

The fluid dynamic of the **Penn State 12 cc** [3] pediatric pneumatic VAD was characterized through an innovative dynamic mesh boundary approach in order to realistically mimic the membrane kinematics and quantitatively capture, better than a pure CFD analysis, the hemodynamics inside the device.

## Materials and Methods

### Simulation of membrane kinematics

A **FE simulation** was conducted on **Abaqus 6.10 (SIMULIA®)** to quantitatively assess systolic membrane deformation and extract its kinematics; at this aim, the time-dependent curve of systolic pressure [4] was uniformly applied on the upper membrane surface (Fig. 2). Nodal displacements of the membrane were then exported through ASCII-formatted files and used as boundary conditions in the subsequent **CFD analysis**. Such a procedure was accomplished implementing a **user-subroutine** in order to update, through time, the mesh of membrane surface.

## CFD Simulation

VAD systole was run through transient **CFD simulation** in **Fluent v13.0 (ANSYS®)**. Blood was characterized as a Newtonian incompressible fluid. A zero pressure was imposed to the surface corresponding to the outlet (aortic valve), the membrane was modeled as a **moving wall** (Fig. 1) while the remaining surfaces were treated as stationary walls. The moving boundary approach, prescribed via user-defined function (UDF) on the membrane and using a Fluent pre-implemented **GRID\_MOTION** macro, was coupled with a dynamic mesh condition applied on the fluid zone adjacent to the membrane. Finally, an adaptive time step was used, thus guaranteeing the stability of the solution and avoiding excessive mesh distortion.

## Results and Discussion

The movement assigned to the membrane well compared with the real behaviour of the membrane as assessed from Particle Image Velocimetry results [5].

A peak of velocity, equal to 18.1 m/s, was computed near the bifurcation of the housing and upstream of the outlet conduit; moreover, blood stagnation phenomena were detected near the inlet inner wall. The contours of velocity magnitude were reported on the three different planes (Fig. 3) at 63 ms of the simulation, i.e. at the instant corresponding to the maximum value of membrane acceleration.

During early systole velocity pathlines were uniformly directed towards the outlet with parallel trajectories and without marked swirling low-velocity fluxes (Fig. 4).

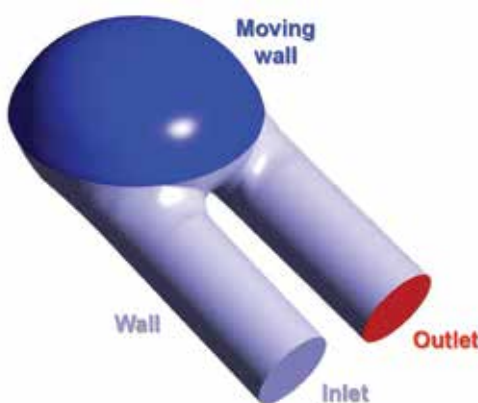


Figure 1 – VAD geometry with CFD boundary conditions

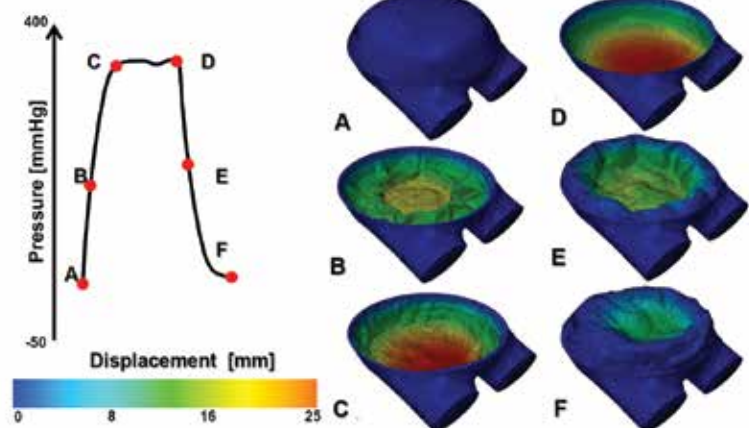


Figure 2 – Membrane configuration at different times and applied systolic pressure

### Conclusions

Integrated FE and CFD simulations allowed to deeper evaluate VAD hemodynamics under realistic working conditions: the proposed approach may improve device optimization; furthermore, it may be used as a valuable numerical tool for thrombogenic analysis in pediatric VAD design.

### References

- [1] D. Lloyd-Jones et al., AHA statistical update (2010) 121:46-215
- [2] C. Long et al., Comput Mech (2013) 52:971-981
- [3] M. Schoenberger et al., ASAIO J (2012) 58:481-493
- [4] B. Roszelle et al., ASAIO J (2010) 56:356-363
- [5] R. Mediviz et al., J Biomech Eng (2009) 131:1-9

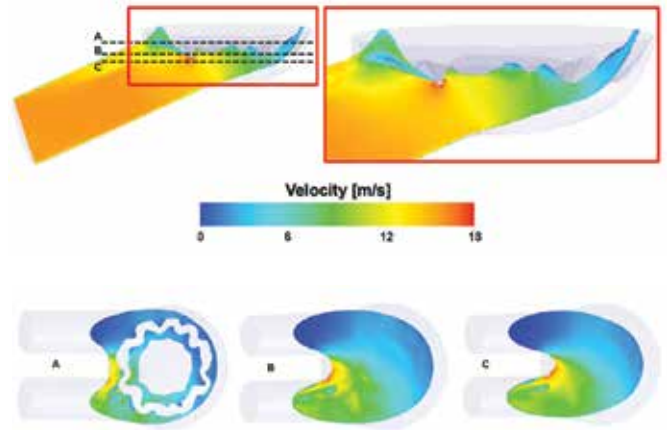


Figure 3 – Contours of Velocity Magnitude [m/s] in longitudinal direction and in three different locations within the VAD at 63 ms

**Alessandro Caimi, Federico Canè, Anna Senesi, Annalisa Dimasi,  
Alberto Redaelli**  
*Dept. of Electronics, Informatics and Bioengineering, Politecnico di Milano*  
**Francesco Sturla**  
*Dept. of Electronics, Informatics and Bioengineering, Politecnico di Milano  
and Division of Cardiovascular Surgery, Università degli Studi di Verona*  
**Keefe B. Manning**  
*Dept of Biomedical Engineering, Penn State University Phipadelphia USA*

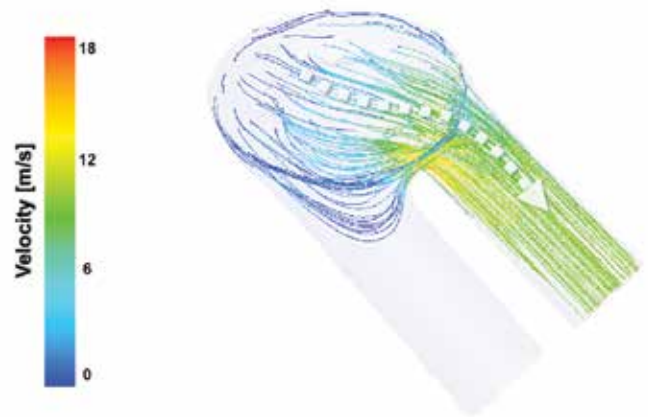


Figure 4 – Velocity pathlines [m/s] at 63 ms





# A fluid-structure approach to optimize the thrombogenic potential of artificial heart valves



## Introduction

Experimental procedures are commonly used to obtain an overall quantification of the fluid dynamic and thrombogenic performances of cardiovascular devices [1].

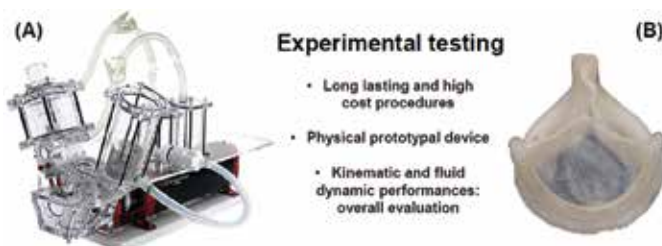


Figure 1– (A) Prototype of the polymeric aortic valve (Innovia LLC, Miami, USA). (B) ViVitro gold standards testing benchmark (ViVitro Labs, BC, Canada)

In this work we have implemented a computational approach to mimic the realistic operative conditions of an experimental set-up and provide extremely localized and detailed information.

Precisely, we presents an innovative tool that emulates a gold standard testing benchmark (Figure 1-A) to perform a micro-scale analysis of the performances of a prorotypal polymeric heart valve (Figure 1-B) in order to identify thrombogenic localized hot-spots for further possible design optimizations.

## Materials and Methods

The computational models (Figure 2, left) and their finite element discretization were implemented on ANSYS Gambit 2.4.6 (ANSYS, Canonsburg, USA). The FSI simulation was performed with the explicit solver LS-DYNA R6 (LSTC, Livermore, USA), coupling eulerian fluid elements and lagragian solid ones.

## Boundary Conditions

The Inlet and Outlet reservoirs were characterized by physiological pressure waveforms extracted from experimental testing, thus obtaining comparable working conditions between the computational model and the ViVitro benchmark.

## Thrombogenic analysis

A particle taking methodology was adopted to obtain the trajectories of numerical ideal platelets. Numerical models were used to combine

stress and time ( $\tau(t)$ ) so as to perform a quantification of the thrombogenic potential of the device as well as of critical hot-spots.

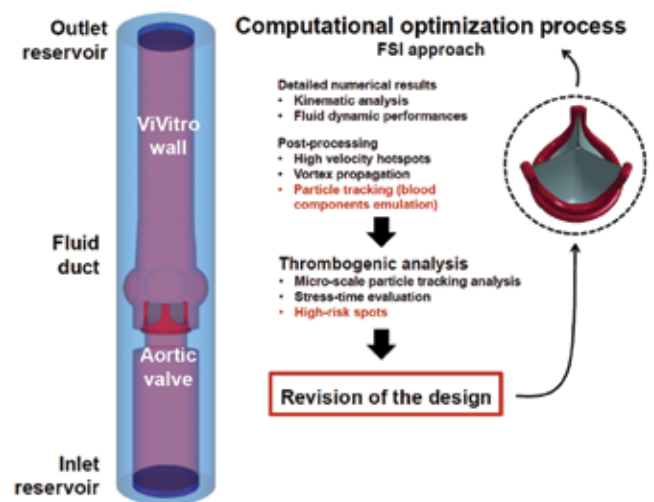


Figure 2 - Schematic visualization of the ideal optimization process implemented with the FSI numerical approach

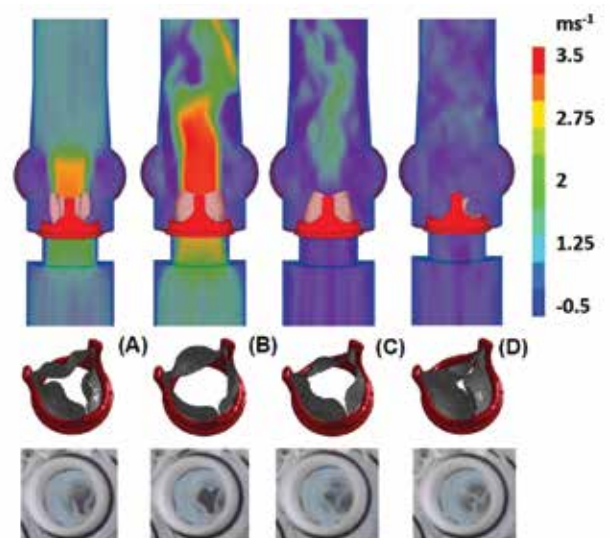


Figure 3 - Visualization of the velocity field contours on a cross-sectional plane (upper panels) and of the FSI and experimental kinematic of the device (lower panels), at four different time-points: (A) early systole, (B) systolic peak, (C) closing phase, (D) stable diastole

	FSI	Exp. [2]	
$V_{max}$	3.37	3.42	$ms^{-1}$
Effective Orifice Area	1.54	1.47	$cm^2$
trans-valvular $\Delta P$	18.60	20.91	mmHg
Ejection time	320	304	ms

Table 1 - Numerical verification variables

### Results and discussion

Primarily, the numerical solution was compared with experimental results and its reliability was verified: as reported in Figure 3, valvular kinematics was qualitatively compared with the in-vitro mechanical response.

Moreover, a quantitative validation of the numerical results was accomplished through the evaluation of fluid dynamic variables and the comparison with device performances, as highlighted in Table 1.

The thrombogenic evaluation was able to highlight the commissural zones as high-risk locations, due to vortexes propagation (figure 4-A). Micro-scale analysis was used to extract the stress-time history of selected dangerous trajectories.

### Conclusions

The proposed tool was able to accurately replicate realistic experimental working conditions. Moreover, a micro-scale analysis provided detailed information regarding the thrombogenesis potential of particular zones, potentially allowing for further design optimization of the device.

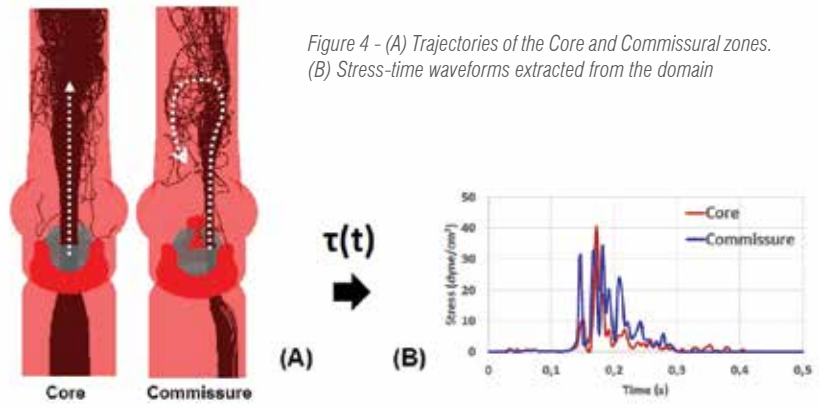


Figure 4 - (A) Trajectories of the Core and Commissural zones. (B) Stress-time waveforms extracted from the domain

### Aknowledgements

The research leading to these results has received fundings from the Cariplo Foundation Project, Grant Agreement N°2001-2241.

### References

1. D. Bluestein et al. (2010) Annals of Biomedical Engineering
2. T.E. Claiborne et al. (2013) Journal of Biomechanical Engineering

*Filippo Piatti, Alberto Redaelli*

*Dept. of Electronics, Information and Bioengineering, Politecnico di Milano*

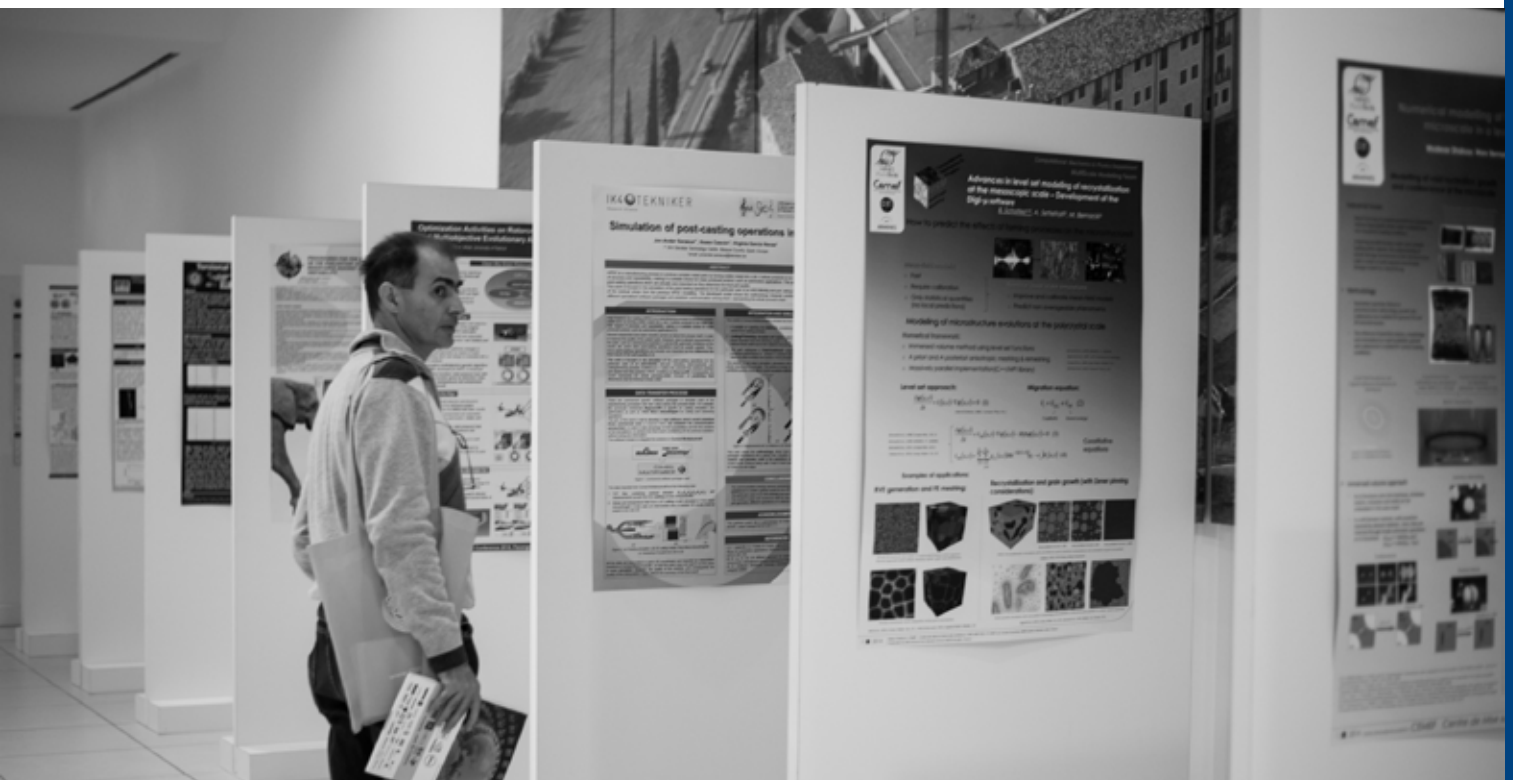
*Francesco Sturla*

*Dept. of Electronics, Information and Bioengineering, Politecnico di Milano*

*Division of Cardiac Surgery, Università degli Studi di Verona*

*Thomas E. Claiborne, Danny Bluestein*

*Dept. of Biomedical Engineering, Stony Brook University, New York, USA*



CAE POSTER AWARD 2014: WINNER

# Enrichment of the Finite Element Method with the Meshless Local Petrov-Galerkin method in Structural Mechanics



UNIVERSITÀ  
DEGLI STUDI  
DI PADOVA



## Introduction

### Finite Element Method

The Finite Element Method (FEM) is a consolidated and reliable approach in computational mechanics. However, in some problems it can suffer from the drawbacks associated with the use of meshes. The creation of a mesh can be both a complex and a computationally expensive task, and in many cases it requires human intervention, or otherwise involves an added programming effort.

### Meshless

To overcome the difficulties related to the use of a grid, several meshfree methods have been proposed. They discretize the equations on scattered clouds of points, or "particles," thus avoiding any topologically connected data set. However, their greater flexibility is counterbalanced by a much higher computational cost that can reduce the range of their engineering applicability.

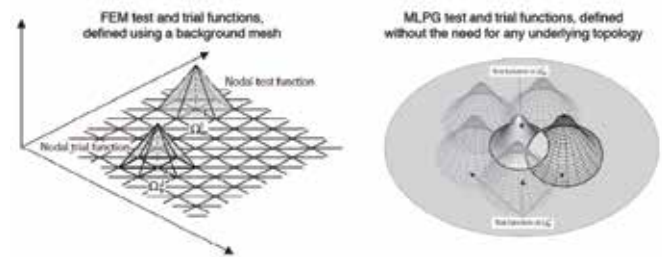
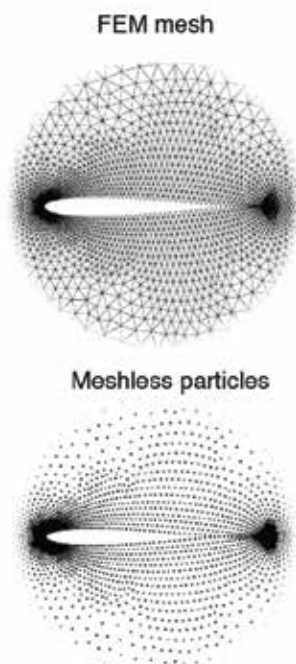
### Aim

#### Improve the FEM solution without remeshing

The main idea of this work is to "enrich" the FEM with a meshfree method, in order to emphasize the merit of each approach. Specifically:

- the superior performance of the Finite Element Method provides the bulk of the computational burden
- the particles are added only in a sub-region of the computational domain, where their greater versatility is needed

The solution can be improved, as the particles are added freely in the area of interest, and with no remeshing cost.



## Enrichment in Structural Mechanics

### A local approach in linear elasticity

The meshless technique employed in this work is the Meshless Local Petrov-Galerkin (MLPG) method [1]. The distinctive feature of MLPG is the lack of any background cell for the energy computation, hence it is a "truly" meshfree approach with a considerable flexibility of use. We notice that the Local Symmetric Weak Form (LSWF), which is the basis of the MLPG approach, encompasses the Galerkin Finite Element

Method, when the test and trial functions are chosen appropriately.

The consideration above suggests to start from the LSWF of the MLPG approach, and then to make a formal distinction between the FEM and MLPG functional spaces. The final system of equations for the enriched method arises quite naturally, as long as both spaces are considered together.

### Fully Coupled Enrichment

#### Two Way Approach

The discretization of a structural problem, by superimposing FEM and MLPG, yields:

$$\begin{bmatrix} \mathbf{K}_{FE} & \mathbf{K}_{MF} \\ \mathbf{K}_{FM} & \mathbf{K}_{ML} \end{bmatrix} \cdot \begin{bmatrix} \mathbf{u}_{FE} \\ \mathbf{u}_{ML} \end{bmatrix} = \begin{bmatrix} \mathbf{b}_{FE} \\ \mathbf{b}_{ML} \end{bmatrix}$$

- The diagonal blocks are the standard FEM and MLPG stiffness matrices
- The vectors on the right hand side are the usual FEM and MLPG forcing terms
- The two off-diagonal blocks are "coupling blocks," as they link the FEM to the MLPG method.



The essential feature of the Fully Coupled approach is that both FEM and MLPG “interact” to form the solution.

**Uncoupled Enrichment**

**One Way Approach**

The final system of equations of the Fully Coupled enrichment is simplified, as outlined in [2], by neglecting the upper right block, thus obtaining two separate FEM and MLPG problems:

$$\begin{bmatrix} \mathbf{K}_{FE} & \mathbf{0} \\ \mathbf{K}_{FM} & \mathbf{K}_{ML} \end{bmatrix} \cdot \begin{bmatrix} \mathbf{u}_{FE} \\ \mathbf{u}_{ML} \end{bmatrix} = \begin{bmatrix} \mathbf{b}_{FE} \\ \mathbf{b}_{ML} \end{bmatrix}$$

$$\begin{cases} \mathbf{K}_{FE}\mathbf{u}_{FE} = \mathbf{b}_{FE} \\ \mathbf{K}_{ML}\mathbf{u}_{ML} = \mathbf{b}_{ML} - \mathbf{K}_{FM}\mathbf{u}_{FE} \end{cases}$$

- The FEM problem is solved first and independently
- A small MLPG problem is then solved to “correct” the FEM solution

This simplified approach can be derived from the LSWF, with the FEM contribution obtained from a previous computation. As compared to a Fully Coupled approach, the Uncoupled enrichment consists of a MLPG contribution added to an existing FEM solution, hence this is a “one way” enrichment.

**Numerical Results**

The proposed methodology is evaluated in a numerical experiment where the analytical solution is known. The plate in figure is subject to uniform farfield stress. The extremely coarse FE mesh is unable to capture the steep gradient of stress in the vicinity of the hole. However, when few particles are added, the solution is improved and the local stress field is accurately represented around the hole.

**Conclusion**

The FEM and the MLPG method are combined in the LSWF framework to obtain a “Fully Coupled enrichment”. A less costly alternative, called “Uncoupled enrichment” can be also obtained. In the latter, only a linear system, as small as the number of added particles, needs to be solved to improve an existing FEM solution. Further refinement by the use of particles requires that only the small MLPG problem be solved again. In our numerical experiments, the addition of a limited number of MLPG nodes has proven to be effective for a local improvement of the solution field, without changing the underlying FEM mesh.

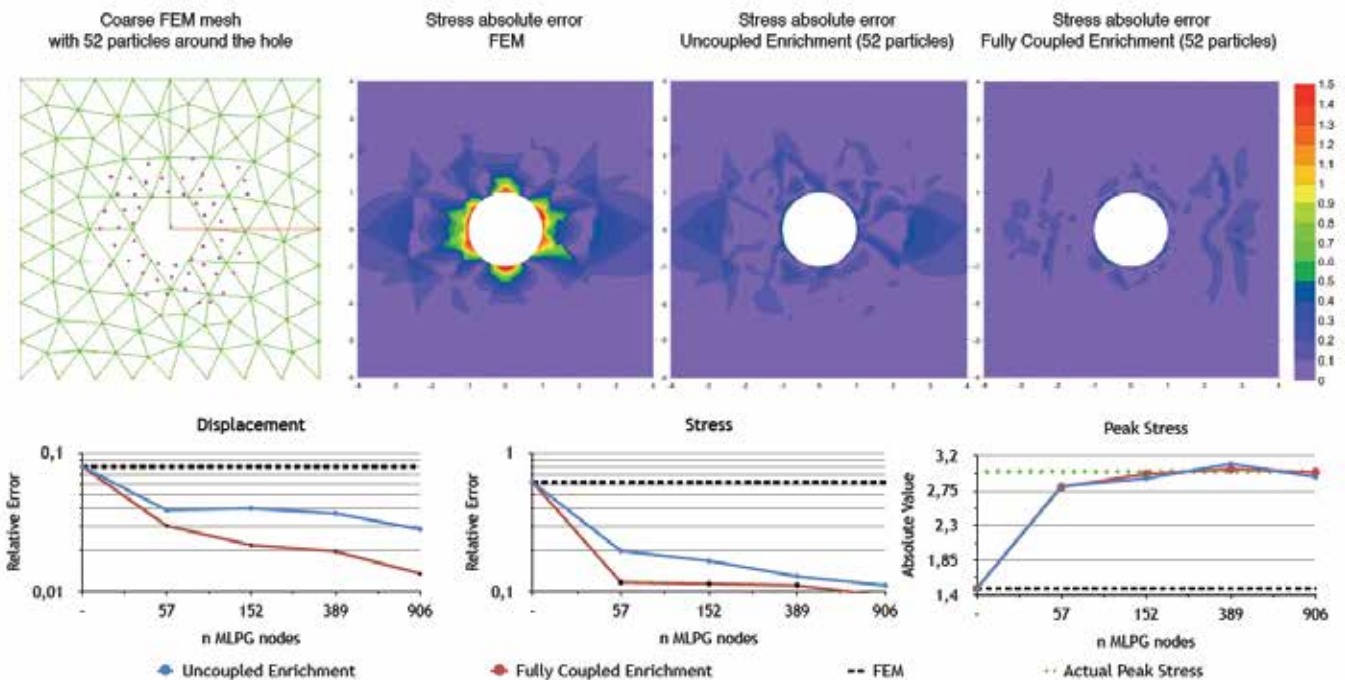
In essence, the present approach appears to possess a potential for improving a Finite Element solution field, in terms of accuracy and flexibility, especially where a local, steep gradient of stress exists, at no remeshing cost.

**Bibliography**

- [1] S. N. Atluri, T.-L. Zhu, The meshless local Petrov-Galerkin (MLPG) approach for solving problems in elastostatics, Computational Mechanics 25, (2000), 169-179
- [2] M. Ferronato, A. Mazzia, G. Pini, A Finite Element Enrichment technique by the Meshless Local Petrov-Galerkin Method, Computer Modelling in Engineering & Sciences, 62, (2010), 205-222

*Andrea Zanette, Massimiliano Ferronato*

Department of Civil, Environmental and Architectural Engineering,  
University of Padua



# Effect of volumetric flow rate on transition to turbulence in patient-specific arteriovenous fistulae for hemodialysis



## Background

Radial-cephalic arteriovenous fistula (AVF) is the best option for vascular access for hemodialysis (HD). The AVF has unacceptable early failure rates due to formation of neointimal hyperplasia, a fibromuscular thickening of the vessel wall, mainly on the juxta-anastomotic vein (JAV) and on the anastomosis floor. Our previous image-based CFD study in four AVF revealed laminar flow in the proximal artery (PA) and transition to turbulence in the JAV [1].

## AIM

To investigate the effect of flow decrease on flow instability in the JAV in a model of patientspecific AVF.

## Methods

### Meshing of the AVF model

To extract the real geometry we used magnetic resonance angiography (MRA) of a subject starting HD, performed 6 weeks postoperatively. 3D reconstruction of the AVF model from the MRA images was performed with vmtk [2] (see Figure 1).

### CFD simulations of blood flow

Patient-specific boundary conditions have been obtained using the python network Solver (pyNS), a 0D/1D pulse wave propagation model to support decision-making in vascular access planning in the clinic [3]. Flow boundary conditions (Dirichlet) have been imposed on the PA and the DA. Tractionfree condition (Neumann) was imposed at V outlet and no-slip at the walls which were considered rigid (see Figure 2).

Blood has been modelled as a Newtonian and incompressible fluid, with a dynamic blood viscosity of 0.024 Poise and density of 1.05 g/cm<sup>3</sup> was assumed. Three CFD simulations were run imposing firstly full (the AVF patient-specific), and then halved and quartered volumetric flow rates in the PA. Flow rates and Reynolds numbers in PA, DA and V are shown in Table 1.

### Data post-processing

Velocity waveforms were analyzed in some feature points identified along the centerline of the model. Reciprocating disturbed flow zones were localized by means of the oscillatory shear index (OSI) [4]. Wall shear stress (WSS) waveforms were analyzed in some points located by high OSI.

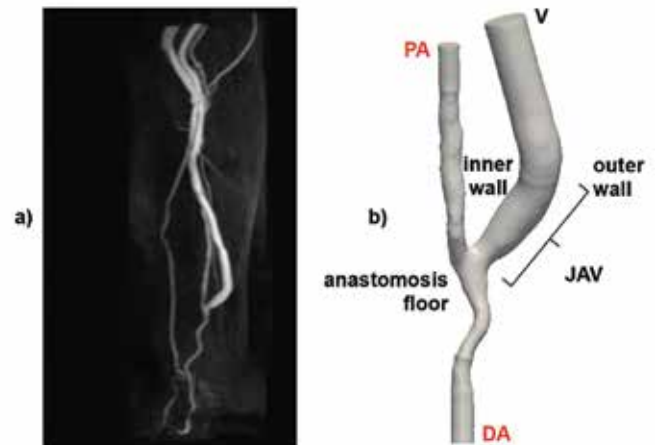


Figure 1. a) Maximum intensity projection (MIP) of the MRA stack showing the radial artery connected to the cephalic vein; b) 3-D surface model of the side-to-end AVF. Legend: PA, proximal artery; DA, distal artery; V, vein

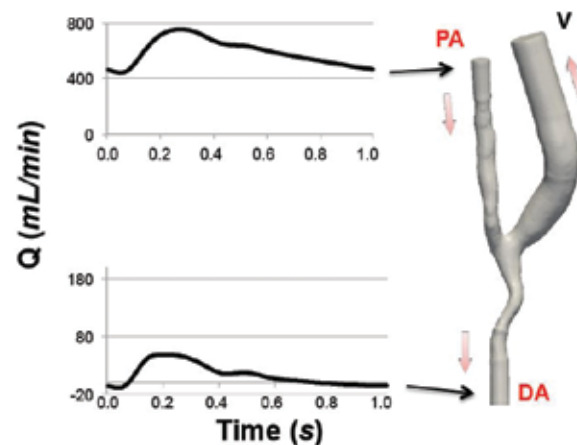


Figure 2. Patient-specific boundary conditions: flow waveform in PA and DA

	$Q_{in}$	$Q_{in} / 2$	$Q_{in} / 4$
$Q_{mean} PA [mL/min]$	591	296	148
$Q_{mean} DA [mL/min]$	13	6	3
$Q_{mean} V [mL/min]$	578	290	144
$Re PA$	1502	751	375
$Re DA$	33	17	8
$Re V$	740	370	183

Table 1. Characteristics of CFD simulations

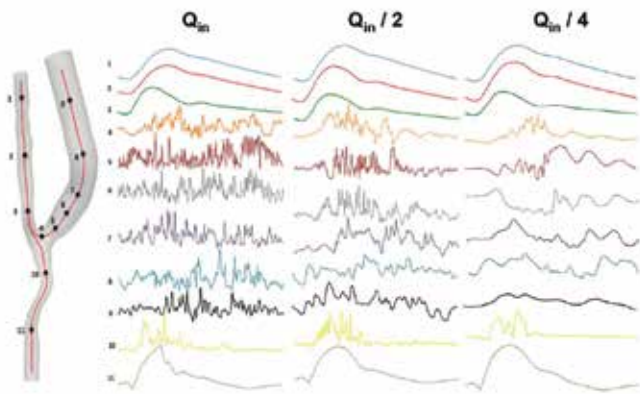


Figure 3. Velocity vs. time traces corresponding to  $Q_{in}$ ,  $Q_{in} / 2$  and  $Q_{in} / 4$

**Results**

*Velocity field*

Velocity, normalized by its cycle-average, revealed laminar flow in the PA and flow instability in the JAV, right after the anastomosis, that damps slightly in the more distal vein. By reducing the volumetric flow rate, velocity instability damps and the transitional flow tends to turn to laminar flow in the vein (see Figure 3).

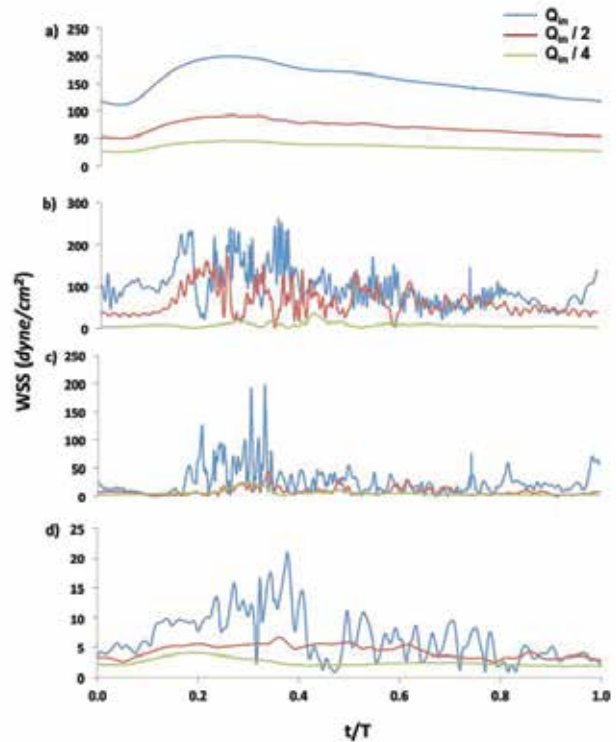


Figure 5. Time traces of WSS magnitude on the period in different sites

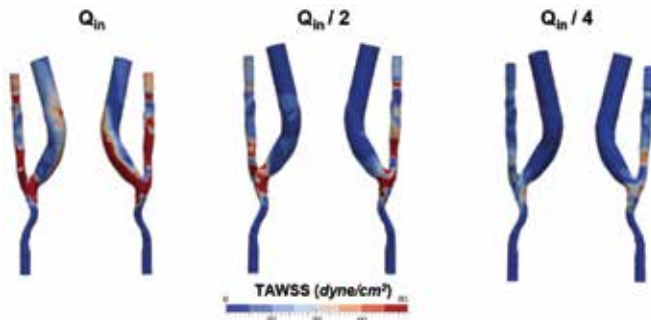


Figure 4. Patterns of TAWSS corresponding to  $Q_{in}$ ,  $Q_{in} / 2$  and  $Q_{in} / 4$

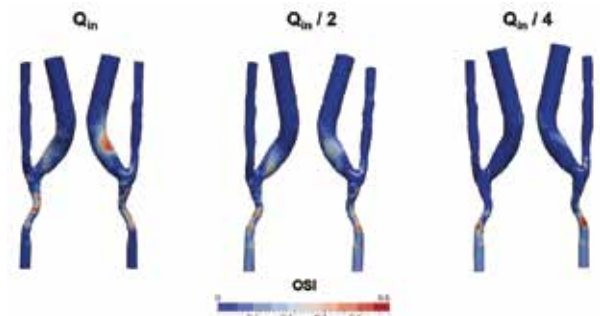


Figure 6. Patterns of OSI corresponding to  $Q_{in}$ ,  $Q_{in} / 2$  and  $Q_{in} / 4$

**Wall shear stress (WSS)**

Time averaged WSS (TAWSS) is presented in Figure 4. Waveform of WSS magnitude in some feature points located by high OSI on the JAV (b and c) and DA (d), and on the PA for comparison with laminar flow (a), revealed damping of the oscillations in the V and DA according to the reduction of flow (see Figure 5).

*Disturbed flow patterns*

By reducing the volumetric flow rate, high OSI zones turned progressively to almost null-OSI zones, revealing a tendency of the WSS to remain in its average direction (see Figure 6).

**Conclusions**

- By reducing the volumetric flow rate, the high flow instability present in the juxtaanastomotic vein has shown a significant damping, towards laminar flow.
- As flow instability has a key role in the neointimal hyperplasia initiation and formation, this finding may help further understanding of the mechanisms responsible for AVF failure.

**References**

1. Remuzzi A, Bozzetto M and Ene-Iordache B. Transitional Flow in Patient-Specific Arteriovenous Fistulae For Hemodialysis. 41st Annual ESAO Congress, Rome, September 17 – 20, 2014.
2. Antiga L, Piccinelli M, Botti L, Ene-Iordache B, Remuzzi A, Steinman DA. An image-based modeling framework for patient-specific computational hemodynamics. *Med Biol Eng Comput.* 46(11): 1097-112, 2008.
3. Manini S, Passera K, Huberts W, Botti L, Antiga L, Remuzzi A. Computational model for simulation of vascular adaptation following vascular access surgery in haemodialysis patients. *Comput Methods Biomech Biomed Engin.* 17(12):1358-67, 2014.
4. He X. & Ku D: Pulsatile flow in the human left coronary artery bifurcation: Average Conditions. *J Biomech Eng.* 118 (1): 74–82, 1996.

*Michela Bozzetto*, IRCCS – Istituto di Ricerche Farmacologiche “Mario Negri” and Politecnico di Milano. *Bogdan Ene-Iordache*, IRCCS – Istituto di Ricerche Farmacologiche “Mario Negri”. *Andrea Remuzzi*, IRCCS – Istituto di Ricerche Farmacologiche “Mario Negri” and University of Bergamo, Department of Industrial Engineering



# 21<sup>st</sup> INTERNATIONAL MASTER'S IN THEORETICAL & PRACTICAL APPLICATION OF FINITE ELEMENT METHOD AND CAE SIMULATION

Due to the next launch of the first professional release of CivilFEM with Marc the students who choose CivilFEM with Marc in the Construction branch will get 50% discount (materials for free)

Since 1993



On-Line Master's

# 2015

Tutorials in  
English & Spanish

Master's Global Partners

UNED

INGECIBER, S.A.

MSC Software

ENGIN  
SOFT

IDAC

## Expert Module

- ⊗ Mechanical Branch
- ⊗ Construction Branch

## Specialized Modules

- ⊗ Dynamic Analysis
- ⊗ Nonlinear Analysis
- ⊗ Heat Transfer Analysis
- ⊗ Advanced Steel Structure Analysis
- ⊗ Fluid Mechanics Analysis
- ⊗ Advanced Concrete Structure Analysis
- ⊗ Geotechnical Analysis
- ⊗ Electromagnetic Analysis
- ⊗ Composite Structures

## Master Module

- ⊗ Final Master's Project

## Software used

### Mechanical Branch

- ⊗ Patran / MSC Nastran
- ⊗ ANSYS

### Construction Branch

- ⊗ CivilFEM with Marc
- ⊗ CivilFEM for ANSYS

### Fluid Mechanics

- ⊗ CFD ++
- ⊗ ANSYS CFD
- ⊗ XFlow

