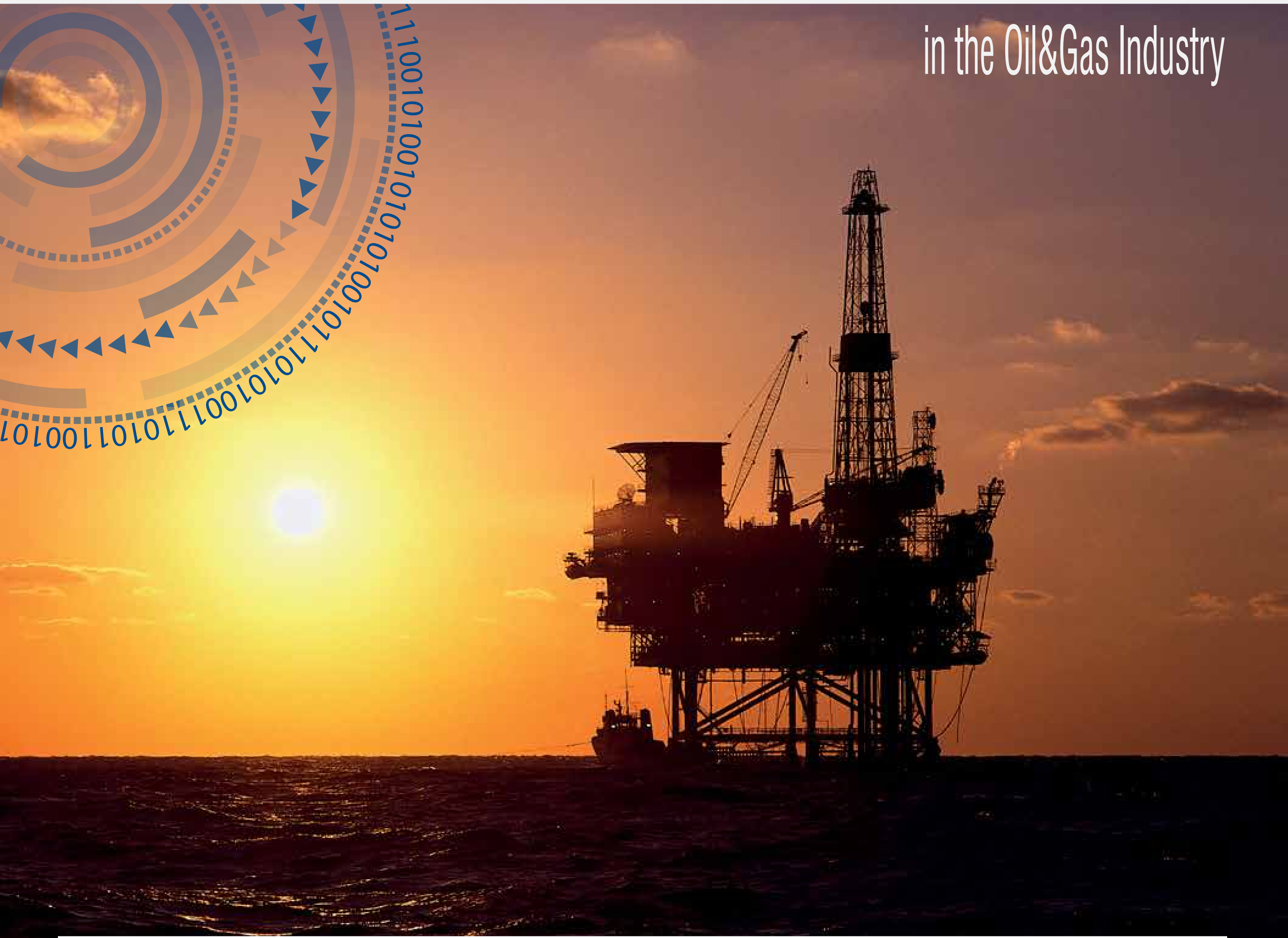
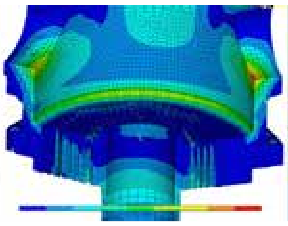
ENGIN SOFT

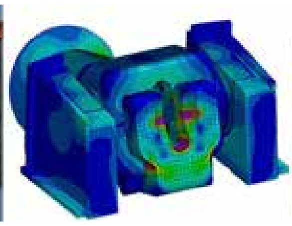
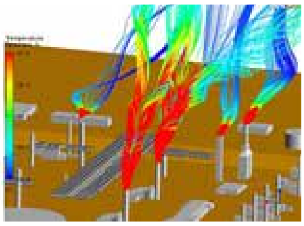
Newsletter

Simulation Based Engineering & Sciences

in the Oil&Gas Industry



CFD analysis of a **lube oil tank**: air ingestion investigation

CO_2 Stripping Column Effluent System Application employing Saipem mechanical innovative Technology

The SuperCups Trays – Saipem's further innovation in **Urea Technology**

System Control logic enhancements through Fluid-Mechanical **valve dynamic transfer** functions

Second **Hydro Power Plant** Turkey is taking off

Evaluation of Surge Pressure Dynamics in a **Closed Fluid Circuit**


EnginSoft Value
Oil&Gas is a mature sector with a long development history and good levels of reliability and performance. Yet change is continuing to accelerate, both in the competitive environment and the development of innovative products.
EnginSoft understands that the demand for continuous improvements cannot be delivered through isolated technologies such as mechanics, fluid dynamics, emerging methods or process calculations. The challenge of Virtual Prototyping is to integrate all of these technologies, and more, into a unique design and optimization process.
This is where the CAE market is moving and this is where EnginSoft are already operating, which is why partnering with EnginSoft can make a real difference to your company. A vast experience in multi-disciplinary simulation consultancy is fueled by continuous investment in in-house R&D and the provision of High Performance Computing (HPC) resources, making EnginSoft the most reliable and knowledgeable partner for product development.
The knowledge and commitment of EnginSoft's engineers' helps clients' identify the most suitable solutions for complex requirements. Leading edge technologies, honed multi-disciplinary methods and access to advanced High Performance Computing (HPC) are all applied to achieve shortened development times and improved resource utilization.
EnginSoft also makes sure that the underpinning expertise is effectively transferred to all customers, who thereby acquire a tangible, strategic advantage.
www.enginsoft.com
The future of conception
to completion
ENGIN
SOFT

Oil&Gas Industry
and the EnginSoft Newsletter Special Issue

The Oil&Gas Industry encompasses a wide range of diverse and demanding environments which poses a varied and continual stream of requirements and problems to solve. These must be addressed both for safe working and retaining competitiveness within the marketplace. From the point of view of engineering simulation the challenges are numerous but they all are characterized by having an important impact on the operating life of an installation or piece of equipment. In tackling a virtual project risk assessment represents an exacting sector requirement and simulation designers must consider on a daily basis the industry regulations that establish criteria for ensuring reliability and safety. Within this complex world of simulation EnginSoft can make all the difference thanks to its globally distributed, multi-disciplinary consultancy and thirty years of hard experience using virtual tools for every conceivable Oil&Gas application. EnginSoft allocates necessary resource through strategic partnerships with major companies such as ESSS, and critically EnginSoft can software engineer the process of simulation. In this context ANSYS is the natural partner where the intelligent fusion of technology and software provides a real difference for continuous improvement with regard to safe working and increased profitability.

Engineering simulation software solutions is critical in enabling Oil&Gas engineers to create virtual prototypes and capture new knowledge about the detailed workings of equipment and processes. Software solutions provided by EnginSoft can apply computational fluid dynamics (CFD), finite element analysis (FEA), and electronic (electromagnetic, electromechanical, signal processing) software to a broad range of drilling, completion, production and processing issues. Diverse benefits include: understanding root cause failures, improving product reliability, evaluating new designs, examining recovery concepts, multiple phase effects, electromechanical effects, particulate motion studies, free surface flows, rock fractionation, erosion and fluid-structure interaction, thermal stresses and soil-pipe interaction.

The powerful realm of simulation can provide detailed maps of parameters critical in optimizing operations and safety such as fluid velocities, pressures, particle concentrations, fatigue, deformation, temperatures, forces and heat/mass fluxes, chemical and species concentration and piping and structural supports. Simulation also opens up new frontiers, such as replication and testing in ultra-deep sea environments or other physically remote terrains, without the high cost and human risk associated

This publication intends to provide the reader an overview of computer aided engineering applications through highlighting EnginSoft's experiences and competencies in multiple technologies, showcasing the benefits of real partnerships via exemplars from ESSS, affirming the core and innovative application of ANSYS technology and finding out what real value was provided to individual customers. Previewed applications include CFD, FEM, CFD 1D, optimization, manufacturing and multi-physics approaches.

As Oil&Gas organisations face ever increasing challenges the relevance of virtual operations lies in their ability to maximise value from existing assets, their usefulness in unlocking barriers to new projects and their proven track record in providing ideas, technologies and innovative solutions to safely control or lower unit costs. As change continues to accelerate we know that adopting and developing simulation based engineering can make a real difference.

Luca Marcadent, EnginSoft
Alistair Murray, EnginSoft UK

For more information or for support in Oil&Gas Industry Simulation
please refer to our dedicated Competence Center:
Luca Marcadent, l.marcadent@enginsoft.it
Phone: +39 049 7705311

Contents

6 ANSYS CFX applications for **chemical and manufacturing industry**

7 The SuperCups Trays – Saipem's further innovation in **Urea Technology**

11 CFD analysis of a **lube oil tank**: air ingestion investigation

14 **TECHNIP KTI**: process engineering innovation

16 **Casappa S.p.A.**: passion, specialization and development

18 **Polimeri Europa**: quality and progress

20 **Ball valves**: DAFRAM's experience

22 FEM Analysis of Ship to Fixed Offshore **Platform Collision**

24 Numerical FE Simulation Applied to Design of **Floating Roof for Oil Storage**

29 **FSO and Shuttle Tanker** in Tandem Configuration Hydrodynamic Analysis Finalized to the Structural Verification of the FSO Mooring System

31 **QCDC connector** design on a FPSO unit

33 Interview to Eng. Paolo Monti **Saipem Energy** – Offshore Division

35 Reliability Analysis of **Offshore Structures** and Inspection Planning

41 Combined 1D & 3D CFD Approach for **GT Ventilation System** analysis

45 A CFD Methodology for Modelling the Cooling Down Process in a 3-Phase **Subsea Separator Tank**

47 A simulation tool for the Dissociation of **Hydrate Plugs** in horizontal line

49 CFD Evaluation of a New Burner Arrangement inside a **Refinery Single-Fired Furnace**

51 ROCKY DEM and ANSYS CFD COUPLING: a powerful tool for **simulating granular-fluid systems**

53 CAE applications for **Valves** in the Oil&Gas sector

55 Second **HYDRO POWER PLANT** Turkey is taking off

57 **CO_2 Stripping Column** Effluent System Application employing Saipem mechanical innovative Technology

67 **Petrovalves**: a continuous effort into development

69 **GE Oil&Gas**: imagination at work

Newsletter EnginSoft
SPECIAL ISSUE ON Oil&Gas

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

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

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

The EnginSoft Newsletter is a quarterly magazine published by EnginSoft SpA

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

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

Cascade Technologies www.cascadetechnologies.com
Reactive Search www.reactive-search.com
SimNumerica www.simnumerica.it
M3E Mathematical Methods and Models for Engineering www.m3eweb.it

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

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

RESPONSIBLE DIRECTOR
Stefano Odorizzi - newsletter@enginsoft.it

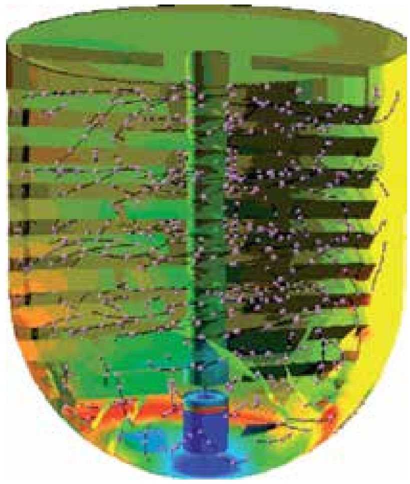
ART DIRECTOR
Luisa Cunico - newsletter@enginsoft.it

PRINTING Grafiche Dal Piaz - Trento

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

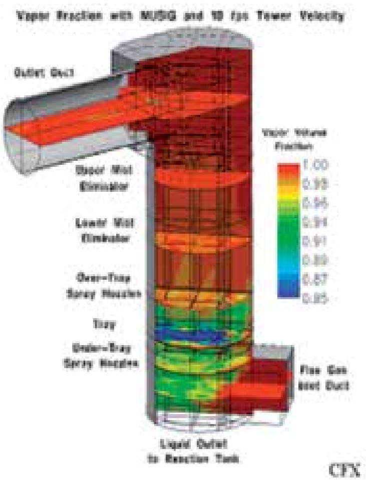
ANSYS CFX applications for chemical and manufacturing industry

The computational fluid dynamics plays today a well-established role in many fields, respect to both development of new products and optimization of already existent ones: the chemical and manufacturing industries are not exceptions.
The commercial code ANSYS-CFX allows to face really complex problems, from different points of view, as geometrical, physical and chemical. The efficiency and simplicity of modeling tools of the package permit to rebuilt the real industrial systems and to produce the FE mesh by using some particularly strong and effective automatic techniques. The direct interface with the most common CAD environments and the use of tetrahedral meshes don't require special geometry manipulation and permit to manage some geometries whose complexity were impossible to approach only few times ago.
Furthermore, the presence of robust numerical models allows to handle the solution of multi-component and multi-phase flows, to study the kinematics of machines' components, rotordynamic machines and to simulate chemical processes with homogenous and heterogeneous reactions.
In ANSYS-CFX these specific models are integrated in a solver for Navier-Stokes equations which has been largely validated and it boasts of the most advanced turbulence models.
The records of industrial problems which can be solved are numerous.
In the field of multi-phase flows the use of the Eulerian-Eulerian and Lagrangian models help to study homogeneous and not homogeneous flows, in which solid, liquid and gaseous phases can be continuous or discrete. These models can simulate the mutual exchange of quantity of motion, energy and mass among the phases. This means that, beyond the simplest transportation phenomena, ANSYS-CFX is able to face problems of thermal exchange both by convention and radiation and it solves phenomena of mass exchange as cavitation, evaporation, condensation and boiling. Among the possible applications there are: mixers, fluid layers, abatement and filtration systems. Moreover open surface flows with eventual solid transport can be solved.
Concerning the rotating machines or machines that allow free body motions two typologies of algorithms are available.
The former is known as Multiple Frame Reference, as it permits to couple both rotating and fixed domains through different coordinate triads. With this algorithm it is possible to run steady and transient analysis. In the first case the forces due to rotations are modeled only numerically, without real motions of the different numerical domains. This approach is valid for different kinds of machines, as pumps, turbines, compressors, mixers, reactors. The transient approach also solves in a more accurate way the same problems coming from the stator-rotor or impeller-dome interactions.
The latter algorithm applicable to machines with mobile parts reproduce the movements through a deformation of the computation grid and it requires the execution of transient simulations. This algorithm is applied for the study of volumetric pumps, development systems, piston systems. For what concerns the chemical processes, mono-phase or multi-phase reacting flows can be studied. The most common applications involve solid, liquid and gaseous combustion phenomena, with specific models for spry, solid fuels (f.e. carbon or biomass) and for black-smoke production. These models allow to simulate burners, furnaces, combustion chambers as also fires with relative problems of smoke dispersion and safety. Moreover the study of reactive flows permits to evaluate the production of pollutants or their abatement and in general any chemical mono-step or multi-step process. In conclusion the development of virtual models through simulation techniques today allows to model the most complex fluid-dynamic phenomena in chemical industry and process industry. Moreover, being the virtual modeling parametric both at geometric and physic level, it permits a fast simulation and exploration of new sceneries.



In this perspective the virtual approach becomes really competitive respect to experimental modeling, as the virtual approach allows to develop and optimize a product/process in short times, reducing the number of real prototypes and determining the guidelines for an effective experimentation. Finally, the know-how acquired thanks to the virtual fluid-dynamics analysis is not based anymore on experience only, but on comprehension of chemical and physical phenomena.

Ing. Lorenzo Bucchieri, EnginSoft



The SuperCups Trays – Saipem's further innovation in Urea Technology

Saipem is a world-leading turnkey contractor in the Oil&Gas industry, providing engineering, procurement and construction services in the design and execution of large-scale offshore and onshore projects, as well as technological competencies for gas monetization and heavy oil exploitation.

One of the main fields of activity of the onshore business unit is the construction of integrated fertilizer complexes to produce ammonia and urea. Saipem has contracted or licensed with its proprietary Snamprogetti™ Urea Technology more than 130 plants for a total urea production exceeding 222,000 metric tons per day all over the world.

The recent Engro (Pakistan) and QAFCO V & VI (Qatar) projects hold the record of the world largest single-line urea units (3850 MTPD), thus overtaking the previous record of the Profertil (Argentina) plant (3250 MTPD).

UREA TECHNOLOGY & INNOVATION OVERVIEW

Urea is nowadays the most widely used fertilizer thanks to its 46% wt nitrogen content which is higher than all other solid nitrogenous fertilizers on the market. Its use as a fertilizer covers more than 90% of its worldwide production. Moreover urea is used also in power plants and diesel engine exhaust systems to reduce emission of NOx gases by selective catalytic and non-catalytic reduction. In the chemical industry it is used instead as a raw material for the production of urea-formaldehyde and melamine-urea-formaldehide resins.

Urea is produced by synthesis from liquid ammonia and gaseous carbon dioxide. In the synthesis reactor (T = 180 - 190 °C, P = 15 - 16 MPaG), the ammonia and carbon dioxide react to form ammonium carbamate (NH_2COONH_4), a portion of which dehydrates to urea (NH_2CONH_2) and water according to the following reactions:

$$2\,NH_3 + CO_2 \leftrightarrow NH_2COONH_4$$
$$NH_2COONH_4 \leftrightarrow NH_2CONH_2 + H_2O$$

The fraction of ammonium carbamate that dehydrates is determined by the ratios of the various reagents, the operating temperature and pressure and the residence time in the reactor.

Urea reactors are Plug-flow reactors ("PFR") type equipped with dedicated distributors for the reagents and a number of sieve type trays ("Standard Design") which consist of perforated plates that prevent back-flow of the heavier solution from the upper part downwards and favour the gas absorption in the liquid phase.

The fluid dynamics of a Urea Reactor can be significantly improved by the introduction of the latest generation of high efficiency trays recently invented and patented by Saipem. The support of a systematic plan of fluid-dynamic simulations gave a significant contribution to the development of the innovative design.

The proprietary Snamprogetti™ SuperCups ("New Design") greatly increases the mixing of the liquid and gaseous phases, respectively ammonia and carbamate, and carbon dioxide, thus optimizing the product conversion rate in the reactor. The immediate benefit is the lower specific steam consumption requirement to decompose carbamate to CO_2 and NH_3 in downstream sections.

This represents a further step ahead to get closest to the theoretical equilibrium conversion in the reactor. In fact, the increase in the reaction conversion is strictly dependent on the mixing conditions of ammonia, carbamate and carbon dioxide through the reactor so that the main purpose of these innovative trays is to further improve the contacting


View of a urea synthesis reactor

conditions among the reagents. Taking into consideration the necessity to minimize the pressure drop across the reactor, the improved mixing is obtained without any increase of compression energy for carbon dioxide.
The innovative concept of the Snamprogetti™ SuperCups lies in the realization of a confined reaction space within the reactor tray geometry, the cups. They perform as a number of mixing units where ammonia is contacted with the gaseous CO_2 in small bubbles.
Once the reactants have swirled inside the cups, the mixed solution of product and non-reacted components is uniformly distributed on the upper part of the tray by means of the upper cup distributor. The outlet flow pattern ensures a further mixing of the solution coming from all the cups.

THE INNOVATION DEVELOPMENT PATH

The development of this innovative solution has been carried out with the support of a comprehensive CFD study. This has been a key tool in the design process from the initial concept idea to the final engineered solution.
Figure 1 shows the multi-step development path followed in the creation of the new design. The blue arrows indicate the work flow directly followed by Saipem while the red-lined arrows refer to the steps where EnginSoft was involved for the CFD simulations.
The driving force for innovation has come from the trend towards increasing plant efficiency with the aim to optimize the capital investment of the equipment, decrease the energy consumption of the high pressure section and therefore reduce the environmental impact of plant operation. It can be seen that the CFD study has been a core activity of the work flow, playing the most important role in the definition of the final design ready for industrial application.
The CFD study first compared the performance of the new tray design with respect to the traditional plates in terms of fluid dynamic behaviour. Subsequently the CFD analysis was used to perform the fine-tuning of the tray design parameters. An iterative approach was applied so that the initial design coming out from the original concept idea (start-up design) was progressively improved until the design parameters provided a satisfactory output.
The iterative "Simulate-Analyse-Redesign" approach delivered a reliable and detailed design which enabled a direct jump to an industrial scale prototype, thus skipping any intermediate phase.
As a consequence, the theoretical design resulting from the simulation study was assessed in the industrial prototype step in terms of mechanical validation, manufacturing activities and maintenance constraints. All the technical problems were solved in order to ensure a feasible and economically convenient construction and to allow an easy installation and maintenance.
In the later step of process test, the developed prototype was then installed in the industrial reactor of a urea producer with the aim to make the performance assessment of the new design.

THE CFD STUDY

Two different geometries were investigated at each step of the iterative CFD study:

- Standard Design: traditional perforated plate (Sieve Trays)
- New Design: innovative plate with perforated cups (SuperCups Trays).

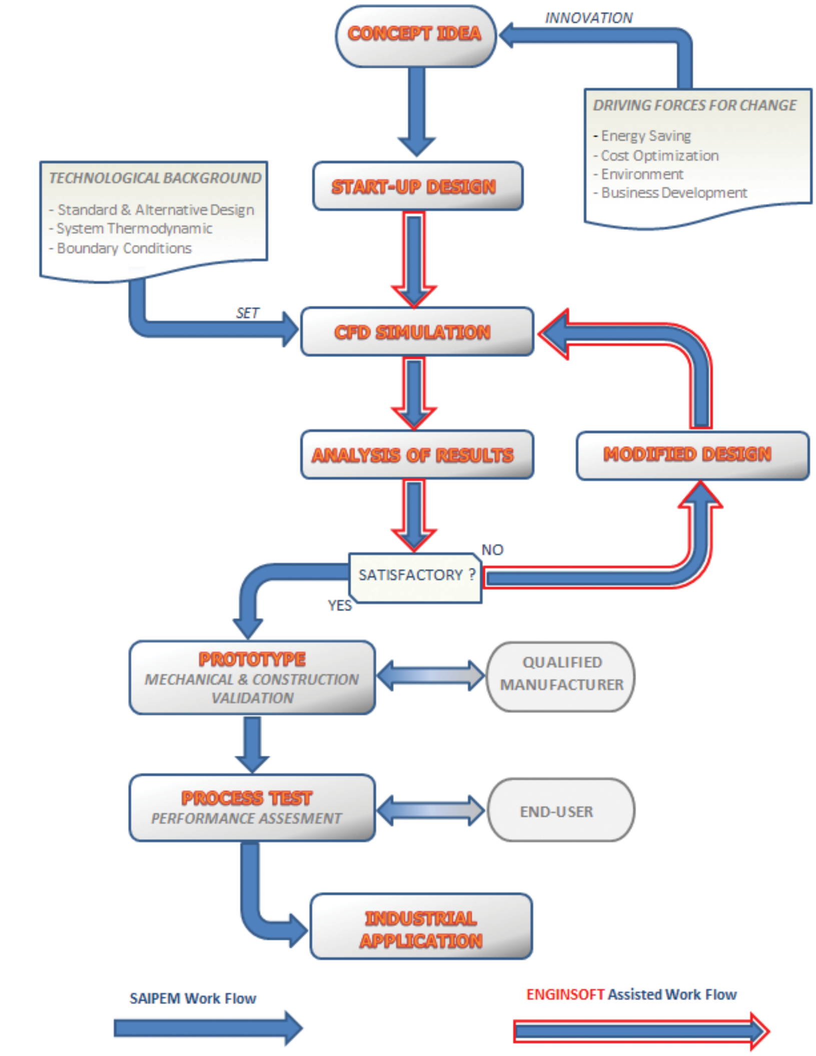


Fig. 1 - Multiple-step Development Path

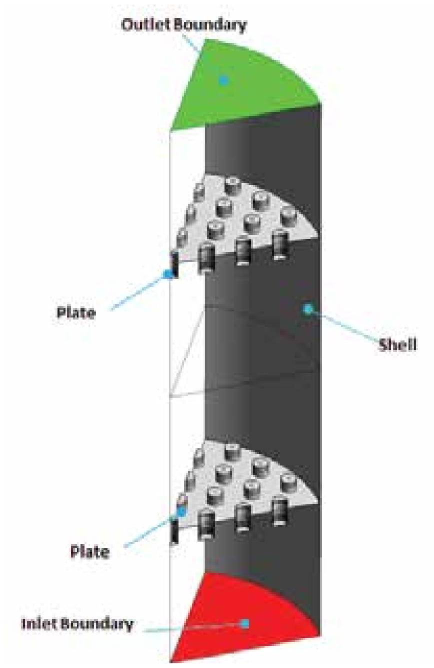


Fig. 2 - Boundary Location for the SuperCups Tray

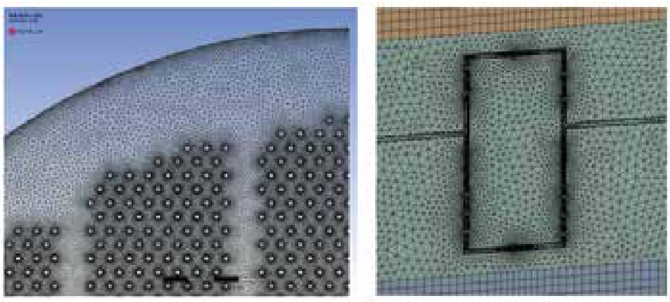


Fig. 3 - Standard Sieve Tray (left) and SuperCups Tray (right): mesh at the surface of the plates

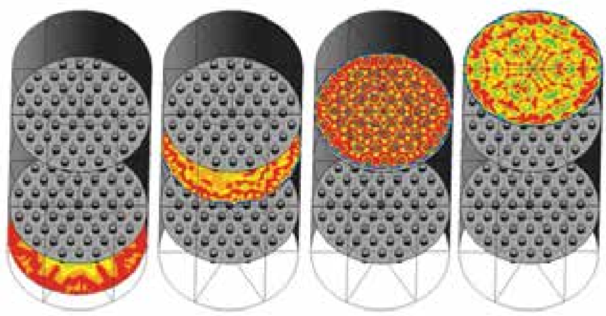


Fig. 4 - CFD analysis of urea reactor

The standard application of sieve trays has been thus used as 'reference point' to which the new design was constantly compared with regard to the mixing degree of the reactants, the flow patterns, the velocity profiles and the pressure drop.

In addition, the Residence Time Distribution (RTD) of each geometry was studied by injecting a pulse input of a tracer in the inlet boundary and monitoring the tracer concentration at the outlet boundary as a function of time.

The software ANSYS-Meshing and ANSYS CFX were used to generate the models and to execute the CFD analyses.

The flow was considered in transient and turbulent conditions. A two-phase non-homogeneous model was used considering NH_3 as continuous phase and CO_2 as dispersed phase. The two phases interact via inter-phase transfer forces (due for example to drag and lift forces exerted by one phase on the other). This aspect and the inclusion of buoyancy allow the calculation of mixing, separation and stratification of the fluids.

The computational domain of each geometry was restricted to the region occupied by two plates. Figure 2 and Figure 3 show the details of the computational domain for the SuperCups Tray and of the mesh for the Sieve and SuperCups Trays (Fig. 2).

The reactants, gaseous CO_2 and liquid NH_3 (plus ammonium carbamate recycle), enter the computational domain from the inlet boundary placed below the plate. The volume fraction of fresh reactants and their velocity profile at the inlet are non-uniform along the radius and are different for the two phases. The main purpose of this non-uniform distribution was to study how the two different designs (Standard Tray and SuperCups) behave when impacted from the same perturbation in different reactor stages and which kind of fluid dynamic patterns develop as a consequence of the arriving wave.

Figure 4 gives a complete picture of NH_3 concentration on the entire geometry of the SuperCups domain at several reactor heights.

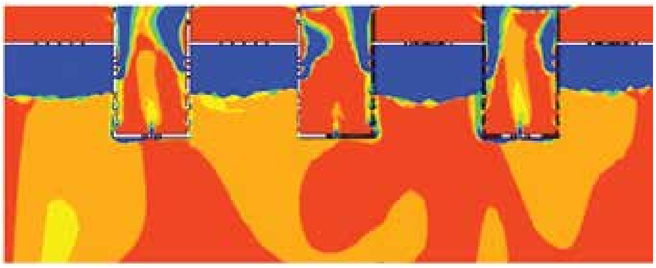


Fig. 5 - First effect: formation of gas cushion

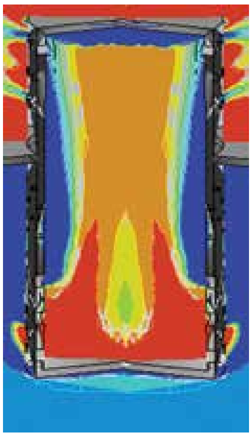


Fig. 6 - Second effect: mixing reactor

RESULTS

The peculiar behaviour of the SuperCups is characterized by a triple fluid-dynamic effect – Gas Equalizer, Mixer Reactor and Gas Distributor – which are described below.

1. Gas Equalizer

The first effect of SuperCups is to uniformly distribute the concentration of the gaseous phase reagent on the entire section of the tray. In this way, the gas bubbles moving upward "lose the memory" of the non-uniformity of the previous reaction stage and the non-reacted CO_2 can be evenly fed to each cup of the tray. Figure 5 shows the formation of the "gas-cushion" (blue area) just below the tray externally to the cups.

2. Mixer Reactor

The cups behave as multiple confined reaction volumes in which the reagents - gaseous CO_2 and liquid ammonia & carbamate – heavily swirl inside, thus reaching a high mixing degree. Each cup performs as a static mixer where the phases are strongly contacted (Figure 6).

In this way the SuperCups Trays do not simply behave as gas distributors – as in other commercial designs – but perform as additional active reaction stages which can be modelled as a Continuous-Stirred-Tank Reactor ("CSTR"). The CSTR behaviour (ideal perfect mixing) of each single tray can be clearly observed by the comparison of RTD curves for the new and standard designs.

Figure 7 refers to the normalized dimensionless RTD curves of the first reaction stage (Tray no. 1). In this case, the mean residence time (Tm) increases by about 70% with respect to the standard design, thus strongly improving the urea formation yield.

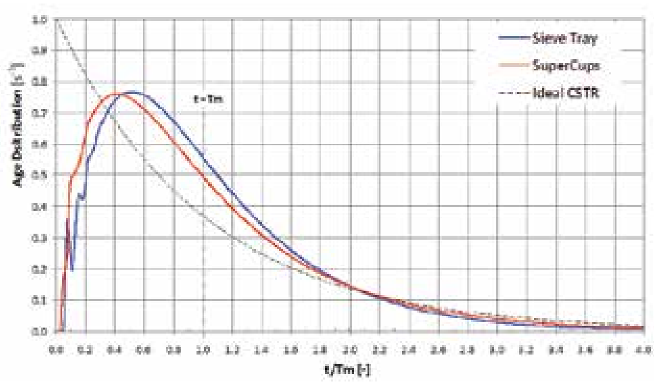


Fig. 7 - Comparison of RTD curves in the first reactor stage (Tray No. 1)

In addition, the curve of SuperCups (red line) better fits the behaviour of ideal CSTR (perfect mixing) than the Sieve Tray (blue line). The application of the "Dispersion Model" (with closed inlet/open outlet vessel boundary conditions) allows the calculation of the Dispersion Coefficient ("D.C.") which is a quantitative measurement of the overall stage mixing. It is found that the D.C. of SuperCups is around 27 % higher than Sieve Tray for the curves of Figure 7.

3. Gas Distributor

The CO_2 gaseous phase forming the gas-cushion below the tray (see Figure 5) can be partially streamed inside the cups to create a mixer-reactor and partially distributed on the upper stage. This split range is one of the most critical design parameter since it allows the customization of the RTD curve of each reactor stage and the increase or decrease of the CSTR (perfect mixing) or PFR (plug flow) behaviour according to the composition of each stage.

CONCLUSIONS

The innovative Snamprogetti™ SuperCups design for Urea reactor trays has been conceived and developed by Saipem with the support of EnginSoft by means of CFD simulation.

The SuperCups Trays permit an increase in the urea reactor efficiency with consequent beneficial effects in terms of higher return on investment, lower energy consumptions and reduced environmental impact.

The CFD study of the traditional perforated plate vs. the innovative tray facilitated the ability to compare the fluid dynamic behavior of several designs in terms of mixing performance of the reactants, flow patterns, pressure drops and residence time.

Indeed, the final result was a reliable detailed design that confidently permitted a direct jump to an industrial scale prototype.

Now on its way to final affirmation, the SuperCups result as one of the main improvements of the latest developments in the Snamprogetti™ Urea Technology.

Ugo Avagliano – Saipem
Massimo Galbiati, EnginSoft

Fig. 8 - First SuperCups installation

CFD analysis of a lube oil tank: air ingestion investigation

Model

A steady state analysis was performed on a General Electric lube oil tank. The aim of the activity was to model the fluid dynamics of the tank during the suction process in order to detect possible air suction by the main lube oil pump.

Figure 1 shows an isometric view of the system. The regions colored in blue correspond to the four inlets through which the oil enters the tank. The pipe colored in green is the discharge pipe of the main pump, while the pipe in red is the discharge pipe of the auxiliary pump. The regions colored in yellow correspond to the suction strainers located upstream of the two pumps.

The model also considers several hollow rectangular bars, two ladders, flanges and a duct for oil filling. There is also a duct for compressor drain located close to central flange.

Due to complexity of the system, a tetrahedral mesh with prism inflation layers was generated with ANSYS ICEM CFD v14. The analyses were performed with ANSYS CFX v14.

Initially, the level of oil inside the tank was at 1.23 m from the tank base. The oil entering from the four inlet pipes transports a small quantity of air. The oil could also enter from a duct representing compressor drain. Two load cases were investigated, considering two different volume fractions of air entering the domain: 0.1 and 0.025. The air could enter only from the two inlets on the right hand side of Figure 1.

Each analysis modeled a three-phase flow where air above the free surface and oil were considered as continuous phases, whilst air entering through the inlets was considered as dispersed gas particles.

The outlet was located at the end of the main pump's discharge pipe. The boundary condition imposed here was an average static pressure equal to 9 bar (pump's head) relative to a reference pressure of 1 atm. The fluid temperature was set to 70 °C. The real walls were modeled as no slip and smooth.

The suction strainers were modeled as porous domains in order to represent the resistance to flow. The pressure loss of the strainers as a function of fluid velocity is reported in Figure 2. The main pump was working, while the auxiliary pump was turned off.

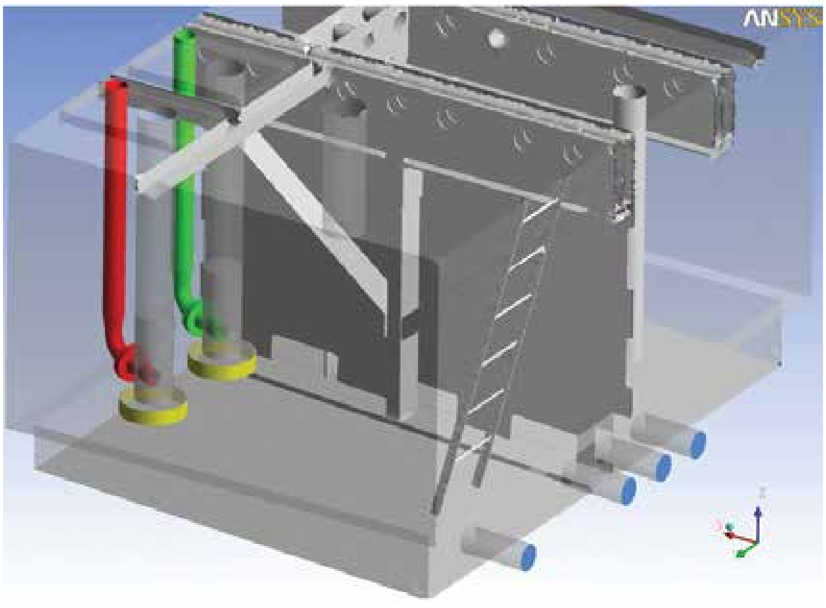


Fig. 1 - Lube oil tank

A buoyancy model was selected with a gravitational acceleration in the negative direction of the z axis. The turbulence model chosen was the Shear Stress Transport (SST) developed to overcome deficiencies in the most common k-ω and k-ε models.

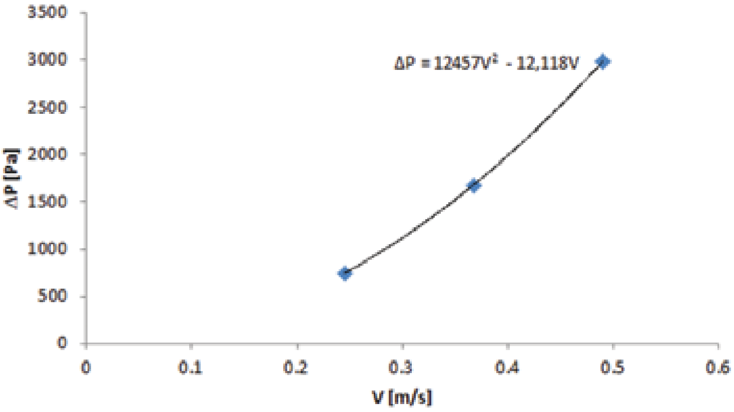


Fig. 2 - Pressure drop of the strainers

Results

The following images show the results obtained considering the two different volume fractions of air entering the domain. Figure 3 shows the oil velocity field in the two cases on a XY plane intersecting the inlets. The maximum value of velocity is reached at inlet 4 because it corresponds to the highest mass flow rate entering the tank. The main difference between the two images refers to Inlet 4. In the case of air volume fraction equal to 0.025 the oil entering from Inlet 4 transports a smaller quantity of air than the case of air volume fraction equal to 0.1 and it can significantly penetrate the mass of oil already inside the tank. As a consequence, the transport of air is more relevant. This phenomenon is highlighted in Figure 4 which reports the contour of volume fraction of air in each case on the previous XY plane. The maximum value of the contour scale corresponds to the volume fraction of air entering from one inlet and therefore it is equal to half of the total volume fraction entering the domain.

Figure 5 and Figure 6 show the iso-surface of volume fraction of air particles leaving the Inlets 3 and 4, respectively. It is clear that air goes towards the free surface as soon as it exits the inlet pipe.

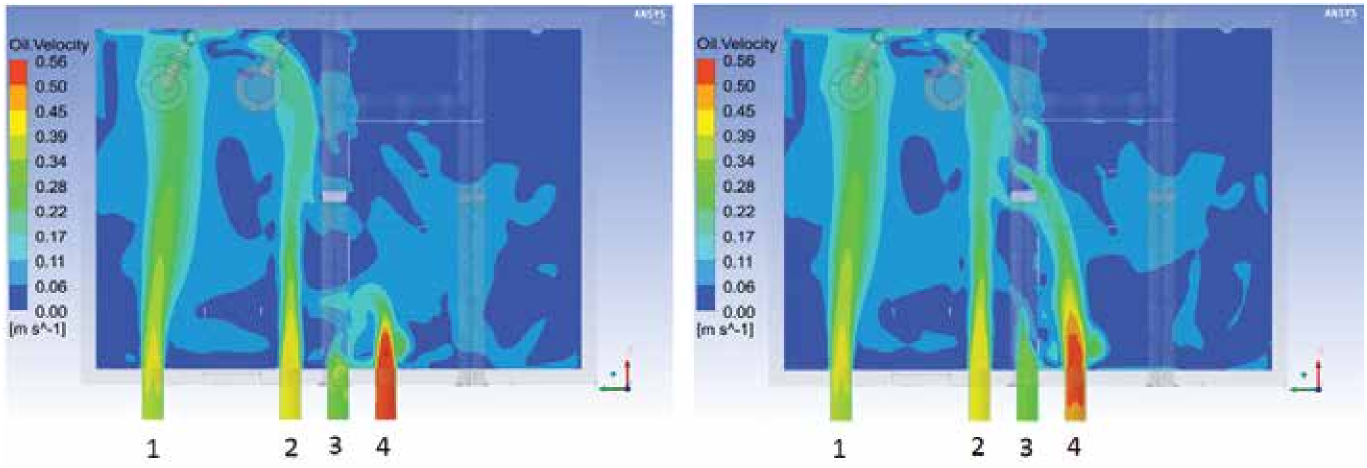


Fig. 3 - Oil velocity field at inlets: with a volume fraction of air entering the domain equal to 0.1 (left) and 0.025 (right)

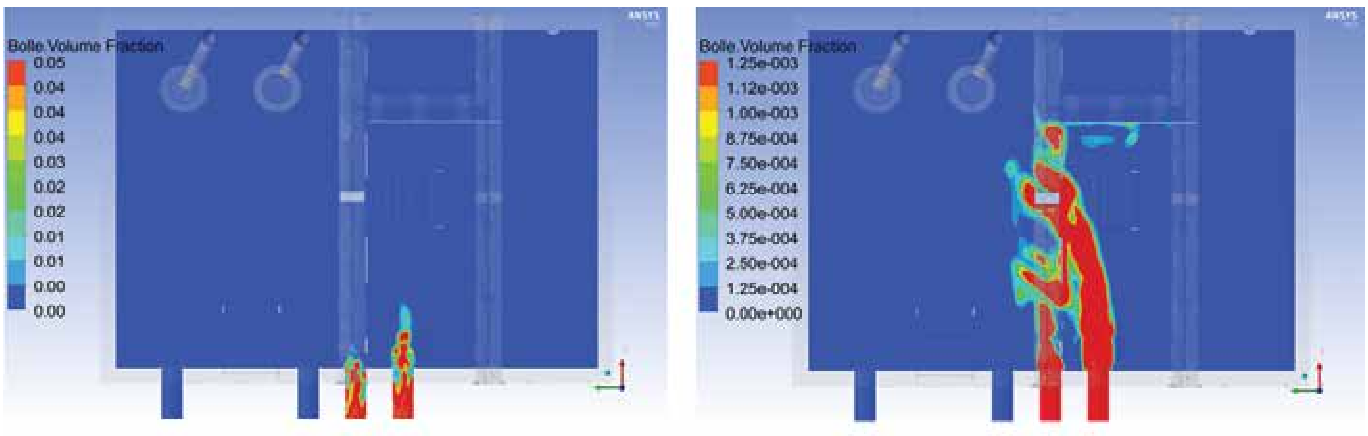


Fig. 4 - Fraction of air particles at inlets: with a volume fraction of air entering the domain equal to 0.1 (left) and 0.025 (right). NOTE: difference in contour scale

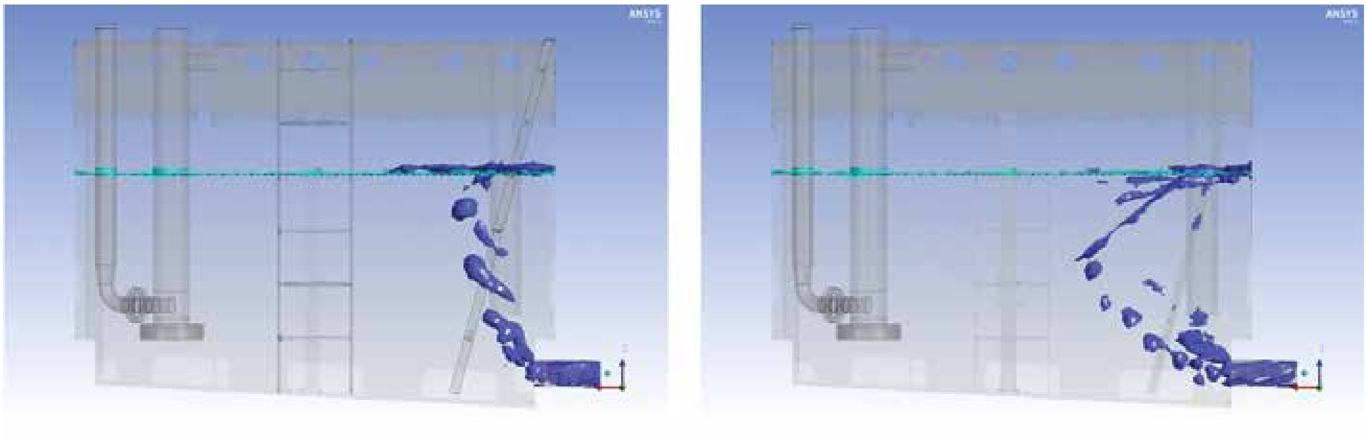


Fig. 5 - Iso-surface of volume fraction of air particles exiting from inlet 3: with a volume fraction of air entering the domain equal to 0.1 (left) and 0.025 (right)

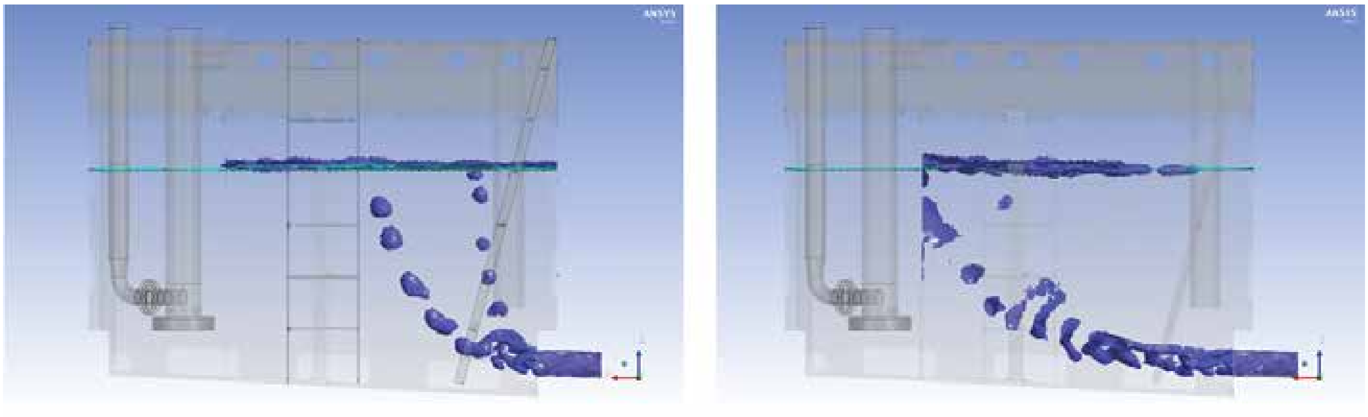


Fig. 6 - Iso-surface of volume fraction of air particles exiting from inlet 4: with a volume fraction of air entering the domain equal to 0.1 (left) and 0.025 (right)

Conclusions

For both the scenarios studied, multiphase analyses shows that there is no air ingestion inside the main pump and all of the air volume fraction entering through Inlet 3 and 4 goes towards the oil-air free surface. As expected, the less the bubble volume fraction entering, the more important becomes the effect of its transport by the oil flow.

Although both analyses were conducted with the auxiliary pump turned off, it is reasonable to consider that no air ingestion would occur even with the opposite scenario, where only the auxiliary pump is working. A similar assumption could not be made were both pumps working simultaneously.

Paola Brambilla, Luca Brugali - EnginSoft
Filippo Viti, Alessandro Pennetta, Daniela Meiattini, Fabien Cochet - General Electric

Inspiring creativeness

Renowned for its performance and inspiring creativeness, GE has always symbolized the idea of progress. Its origins can be traced back to 1878, when Thomas A. Edison founded the Edison Electric Light Company and invented the systems to bring light and electricity all over the world. GE is nowadays operating in extremely diversified technological sectors such as media and financial services, focused on the creation of new products able to improve the quality of everyday life. From aeronautic engines to power generation, covering also sectors like finance, diagnosis, TV programs and plastic materials, GE operates in over 100 countries in the world with about 300.000 employees. www.ge.com

TECHNIP KTI: process engineering innovation

TECHNIP-COFLEXIP industrial group, with a staff of over 18,000 employees, is one of the five main worldwide companies providing process engineering services for petrochemical, chemical, cement and power plants and the like. TECHNIP KTI S.p.A., headquartered in Rome, is the technological centre of the TECHNIP-COFLEXIPGroup. TECHNIP KTI provides qualified services and cutting-edge technologies, as engineering projects, feasibility studies, project management and support, project startup with knowledge transfer. The company's core activities and services mainly are: construction of furnaces and heat exchangers, hydrogen production plants, gas treatment, refuse incineration plants, conservation of energy, combustion systems, environmental engineering.

TECHNIP KTI can boast a well-established experience, with a constant technological growth, oriented to the development of innovative solutions. TECHNIPKTI experts are qualified to follow the overall project, with technical tools at their disposal to support the wide range of engineering disciplines necessary for its carrying out. TECHNIPKTI has invested resources and energies to equip itself with cutting-edge technological systems and to develop an in-house software for project carrying out, management and control. TECHNIP KTI start-up group finalizes the knowledge transfer, providing the customer with operative training, technical assistance and after-sales support. Since 1993 TECHNIP KTI has been certified with the ISO 9001 and it guarantees that services and production

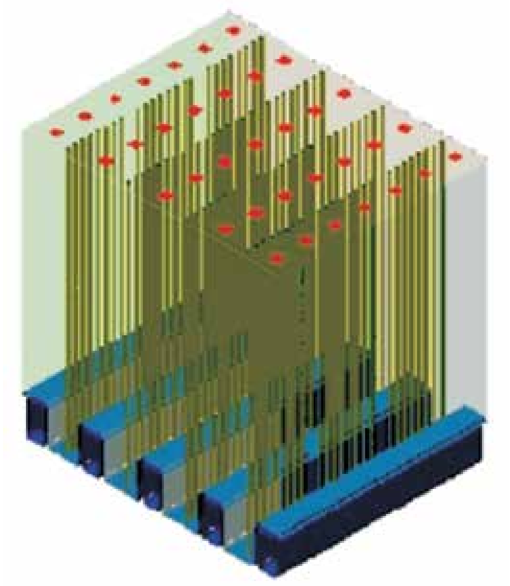
"The use of the ANSYS CFX code has enabled us to slightly reduce the product development time. After a pilot project and in-depth technical and economic evaluation, ANSYS CFX turned out to be the best software. According to our experience, we can estimate a ROI ranging around 6 months; furthermore its modular solutions will permit to face more and more challenging analyses in the future".

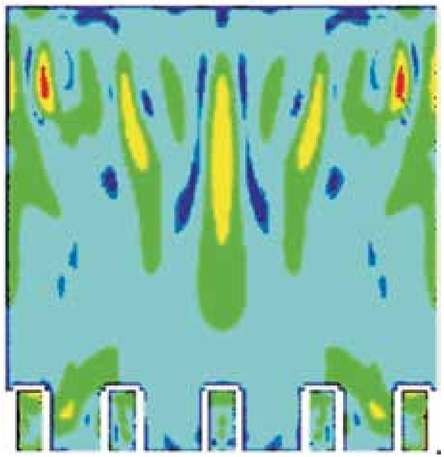
Eng. Ranalli, KTI Technical Manager

"Using ANSYS CFX has proved to be much easier than what we expected, the user interface is extremely easy and user-friendly and the computation code is a really powerful and reliable one. EnginSoft has demonstrated to have really qualified experts that supported us carefully in the delicate of transferring knowledge to our staff".

Ing. Antonelli, KTI Analyses Manager

follow a controlled process, which is oriented to the achievement of the maximum quality so to constantly incorporate innovation into the process engineering.

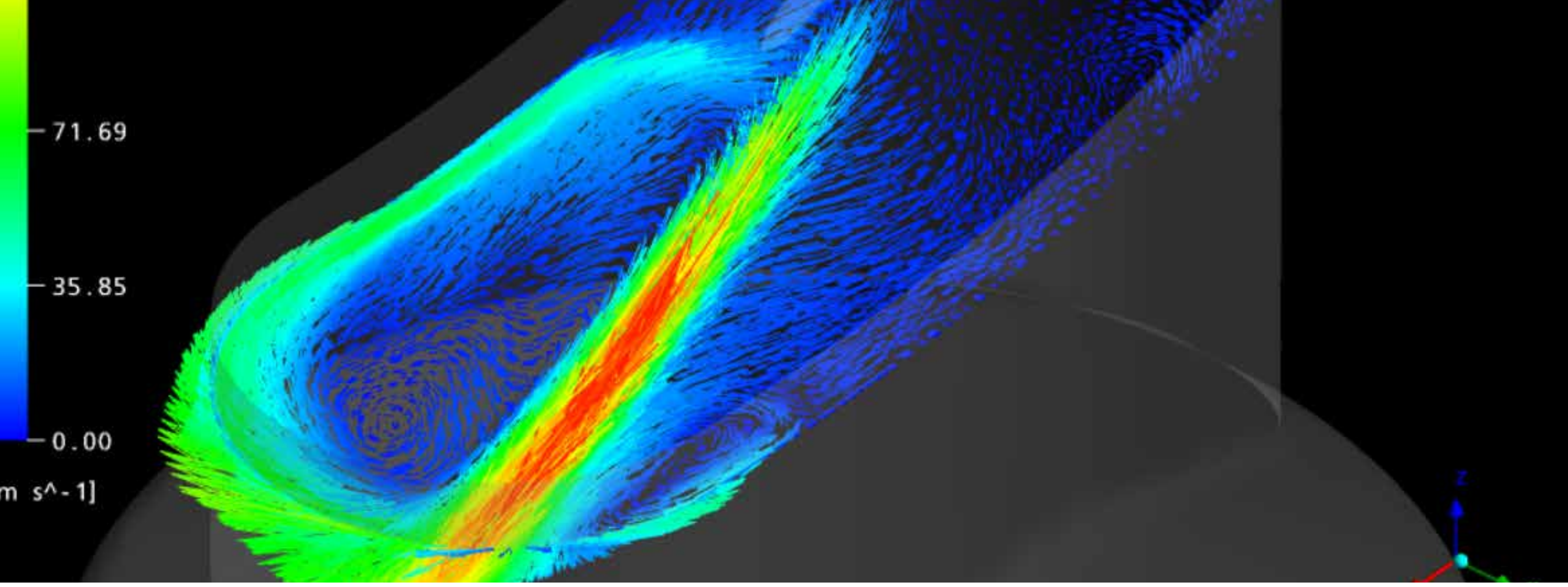




Using ANSYS CFX in design

In order to face at best the analysis phase within the construction project of furnaces, reformers and burners, it is essential to have an in-depth knowledge of the thermal-fluid-dynamic aspects involved. The ANSYS CFX solver, coupled with AMG techniques (something unique in commercial software), allows to easily get the solution and the convergence of the most complex problems, as far as both geometrical and physic-mathematical problems are concerned.

The application of the ANSYS CFX technology has permitted for instance the verification of the capacities and the associated unbalances within reforming tunnels for fumes disposal. It has allowed also KTI designers to achieve a critical vision of the fluid-dynamic field and to make some changes and improvements in it. Other advanced enhancements of ANSYS CFX computation capacity, as combustion and radiation, enable a correct estimation of the flame kinematics, determining the thermal distribution within the burners, together with a computation of noxious emissions. The thermal exchange includes the possibility of heat transfer through radiation, notably favoring the evaluation of the thermal process in furnaces and the like.



Casappa S.p.A.: passion, specialization and development

CASAPPA
FLUID POWER DESIGN

Constant evolution and passion for hydraulics; this has been the key to turning the Casappa of 1952, a manufacturer of hydraulic pumps, to today's business enterprise which, through the holding company, Finrel S.p.A., (controlled by the Casappa brothers), directs five subsidiary companies, 3 joint ventures and 4 associated companies.

The group has more than 900 employees and had a turnover of 176 million euros in 2005. Thanks to the expertise and skills of its workforce and to investments made in research and development, Casappa is able to design and produce the main hydraulic components for applications ranging from construction, industrial vehicles and material handling to agriculture and industry. Besides the supply of standard products, Casappa has chosen to maintain the flexibility required for ad hoc projects, also in small quantities; it is well aware of the importance of being able to closely follow its customers and find timely solutions to their various needs, all of which represent a real stimulus for technological innovation. Having received ISO certification from the British Standard Institution in 1994, Casappa has subsequently obtained Standard ISO 9001:2000 certification, further proof of the quality and professionalism that has always

"Casappa has always regarded technology improvement as necessary to support design. In this sense it has paid a special attention to simulation. Combining its enthusiastic spirit and its critical and careful aptitude for innovation, Casappa has finally decided to invest in CFD simulation. Our test stands and first-class experimentation provide useful information, but they require time-consuming tests without achieving the details, that I could personally observe in CFX Post 3D. Besides the close correspondence between CFX predictions and the physical prototypes, I was particularly impressed by the easy 3D identification of the areas characterized by different possible cavitation problems and the consequent erosion. Once the analyses have been performed, it is outstanding to see how easily the phenomena can be visualized, in flexible ways and also from previously unconsidered points of view. It is therefore an excellent tool for in-depth studies and constructive collaboration. CAE and experimentation are becoming more and more important and complementary tools in Casappa, with the aim of increasing the company know-how, as competitiveness is the possibility of creating innovative and high quality products.We have been positively impressed by EnginSoft qualified professionalism, not just in terms of technical skills, but also considering its accurate know-how transfer, according to our specific needs".

Mr. Marco Guidetti
R&D Manager - Casappa

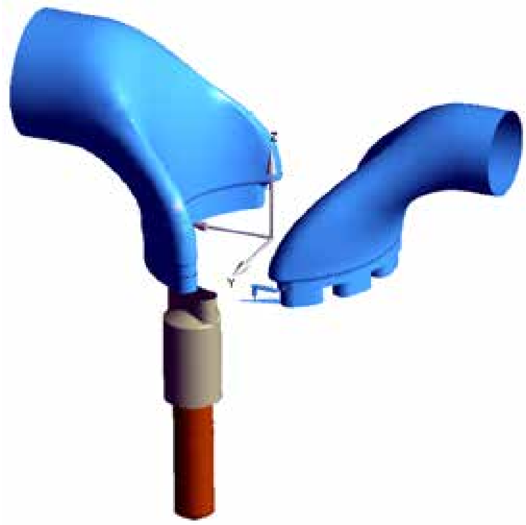
characterized its operations. More recently, Casappa's evolution has been marked by a growing attention to the most advanced virtual prototyping technologies (CAE), also witnessed by its fruitful experience with EnginSoft.

Using ANSYS in Design

The following analysis presents a detailed study of what really happens within a positive displacement piston pump during the complex intake-discharge transient. There is no experimental technique allowing to do this and just a short time ago a reliable CAE methodology was worked out, providing some results in a reasonable time.

First of all, a new reliable physical-numerical model had to be set up, able to simulate such transients, considering gauge pressure changes of at least 280 bar, which normally last just few milliseconds. The first step was the creation of a 3D geometry and of a computation grid, being it an essential element to obtain successful analyses but also a time-consuming operation unless some advanced functionalities are available, such as: immediate robust information exchange with 3D CAD, automatic and adaptive meshes, Generalized Grid Interface (GGI) interpolation techniques, to be clearly understandable to the user. Without these conditions even the construction of the model would be almost impossible, having to deal with a geometry that rotates while the piston has a reciprocating motion. A further fundamental element to be considered was an easy insertion into the code of the fluid characteristic curves, which had been obtained through experimental test, and in particular of the variability of both the compressibility (neither ideal nor null) and viscosity as a function of pressure, as well as the presence of advanced cavitation models working with mobile meshes.

In order to obtain a temporal suitable solution of the analyses, a time step of microseconds and a second order numerical scheme were set up both in space and time, allowing to have a sort of 3D movie made of 4 frames for each angular position degree of the pump group. The computation was run on a first generation Linux cluster at the beginning of 2006, permitting to contain the response time within 24 hours. More recent tests on a second generation cluster have proved how also limited investments in hardware equipment can reduce the computation time to few hours.



Polimeri Europa: quality and progress

Polimeri Europa, a petro-chemical company owned and controlled by Eni S.p.A. , manages through its three divisions (Basic Chemicals, Polyethylene, Elastomers and Styrenics) the production and marketing of a wide portfolio of petrochemical products, having the possibility to use a range of proprietary technologies and state-of-the-art production systems, a wide-reaching and efficient distribution network. Considering its turnover, production volume and people employed, Polimeri Europa is the Italian most important chemical company, holding notable market shares on a European level. Polimeri Europa has its strength in integration: from raw materials to production plants, from research laboratories to technology, through the interface with the market, which can now turn to a single source, with the certainty of finding solutions to its requirements not only in terms of products but also in terms of service, assistance and innovation. This strength is enhanced by its constant commitment to quality and a sustainable development for the environment and the community, formalized and certified in compliance with UNI EN ISO 9001, UNI EN ISO 14001 and EMAS regulations.

Polimeri Europa can also offer its proprietary production technologies, aiming at meeting specific customers' needs. A constant commitment towards

"The increasing improvement in efficiency and reliability of productive processes and product quality is an essential element for a petrochemical company which operates on the European market, dealing with more and more expensive raw materials and energy, as well as with the productive competition coming from Middle and Far East. For this reason in 1998 we started using CFD tools for our R&D activities, and after a thorough analysis of the technologies and the suppliers available on the market, we chose the excellence alliance of EnginSoft and ANSYS."

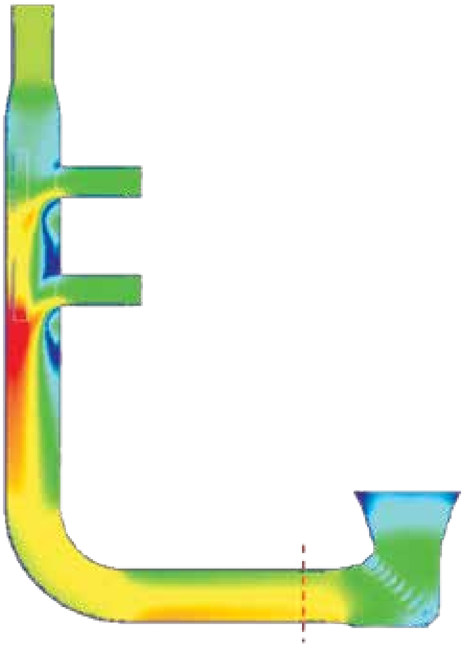
"EnginSoft, thanks to its multi-disciplinary expertise, could guarantee a correct methodology transfer; ANSYS, beside boasting a well-established positioning in this sector, could provide solutions that could be easily modulated, customized and adjusted to any further internal development. After eight years of constant application and thanks to EnginSoft qualified training support, that enabled us to create our indispensable know-how, CFD computation has now become an essential design tool, which is always used to achieve our objectives in terms of product and technology optimization".

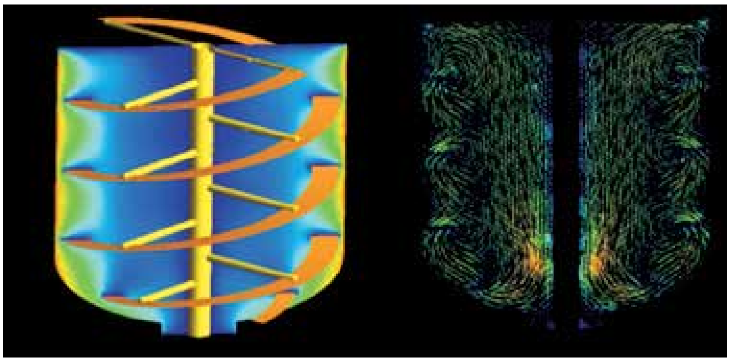
Engineering Manager of Elastomers and Styrenics divisionPolimeri Europa

product, process and technology innovation is the fundamental premise behind strengthening and maintaining Polimeri Europa's competitiveness and leadership. In order to achieve these objectives, Polimeri Europa makes use of a team of about 600 researchers and technologists, distributed in its Research Centres. All these structures are equipped with complex scientific tools that enable them to gain an in-depth knowledge of the product features and to evaluate their performance, so to better meet their customers' requirements. Besides its own structures, Polimeri Europa can count on a wide-ranging network of contacts, as well as co-operations with important university research laboratories both in Italy and abroad. Such structures are highly specialized in different business sectors, dedicated to processes and creating innovative products according to the requirements
of the market. The researchers' active role in the scientific areas of interest, the efficient monitoring of the evolution of product and application technologies, the constant and direct relationship with the customers - also supported by technical assistance structures - allow to focus on innovative objectives and product and process evolution.
Quality and progress. That's the way in which Polimeri Europa presents itself to and can compete on the market.

Using ANSYS CFX in design

Among state-of-the-art CFD solvers, ANSYS CFX has gained a leading position thanks to the wide range of physical modes available, as well as to its easy interfacing and data transferring to other 3D and computing software and its solid and robust algebraic solver. All these features justify
ANSYS CFX application in the petrochemical industry which is particularly oriented to commingling studies, high-speed gas distribution, performance evaluation of mono and multiphase chemical reactors, containing more or less complex rheology fluids and interested in the solution of thermo-mechanical problems regarding industrial equipment. The software recent enhancements allow new applications, still unthinkable few years ago. In particular, the possibility of parallelizing the computation on any HW and SW architecture allows to have a turnaround time, up to the number of CPUs in use and therefore to achieve a computing time speed-up of one order of magnitude.
Moreover the possibility of simultaneously using tetrahedral and hexahedral meshes enables to simulate more and more accurate geometries, while the implementation of LES models permits
a detailed study of the physical phenomena associated with turbulence.





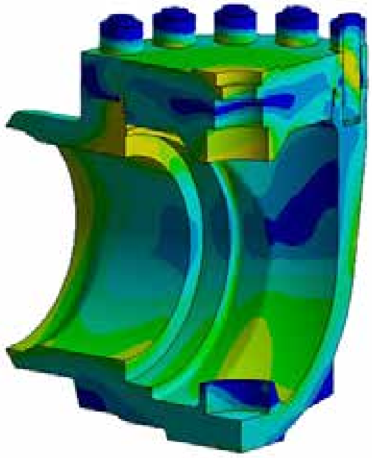
Ball valves: DAFRAM's experience

DAFRAM S.p.A., founded in 1956, was the first company to manufacture floating ball valves in Italy. The long experience gathered during more than 50 years of activity, looking for innovative solutions, ensures DAFRAM as one of the most famous and competitive companies in the world, working in the Oil& Gas sector.

DAFRAM's factory is located in Urbisaglia (Macerata), in the centre of Marche region, in an industrial complex covering 32.000 square meters, 12.000 of which are covered workshops. The factory consists of commercial, technical and engineering offices and of two extremely modern workshops, the latest of which, 4200 square meters, 10 meters high, completed in February 2008, allows the production, assembly, testing, sandblasting and painting of ball valves up to extremely large sizes and weights.
The DAFRAM design and production staff includes highly qualified engineers with a long experience in all technical standards and in meeting customer's special requirements. Modern design methods are employed to analyze specific stresses and deformation limits of valve bodies and main valve components.
The manufacturing process is continuously improved and changed using the most advanced manufacturing technologies such as: multi-function machining centres and several computer controlled lathes and automatized welding equipment. Special testing centers are used for testing all products and are specifically used for high pressure and large size trunnion mounted valves.
In the last years, ball valves have experienced a growing diffusion in the Oil&Gas plants, thanks to a continuous improvement in terms of reliability and

"In the current scenario of the Oil& Gas sector, it is fundamental to combine experience and the development of new solutions focused on operations under the most extreme conditions. DAFRAM as producer of ball valves, in its several versions, has in ANSYS Workbench an efficient and reliable tool to validate and optimize the constructive solutions. This aspect directly affects the reduction of design and production time and costs, that allow the company to become much more competitive".

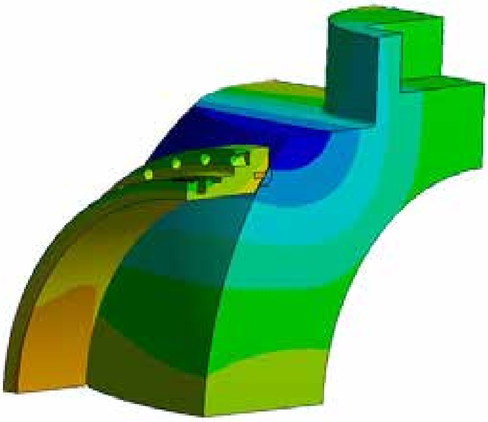
Eng. Marco Sparisci
Product Technical Development
Manager, DAFRAM Spa



performance. If, on one hand, the process conditions (temperature, pressure, corrosive fluids) are becoming more and more stressing, therefore affecting the project requirements, on the other hand, the constructive solutions which have been developed, have allowed the ball valve to become the ideal solution to cope with the interception and the flow regulation also of polyphase fluids.
In this context, design and prototyping activities are fundamentally required to verify and virtually test the designed valve in relation to the real process conditions, considering pressure, temperature and interaction with other plant devices and equipment (actuators, pipes and supports).
Another important application has considered the valve or the whole valve-actuator system under the limit conditions required by the project specifications (seismic analyses or explosion tests) or under specific tests, as for instance hyperbaric one (for submarine valves) or the cryogenic one, that will be experimentally performed by the Testing Department. The optimization and standardization activities of the components take considerable advantage from the virtual prototyping, thus obtaining relevant benefits in terms of weight, reduction of direct costs of the material, of indirect costs of manufacturing, transportation and installing.

ANSYS Workbench for design

DAFRAM Engineering Division has been working to innovate, enhance and improve its products using forefront software tools. Among them ANSYS Workbench is integrated with 3-D modeling software already used for years. The problems under investigation range from structural analysis of the case under pressure (bodies, seals, flanges) and of components controlling the flow regulation (ball, seat, steams) till analysis of the ball-seal tightness; from fluid-dynamic analysis of regulation ball valves, to seismic analysis under specific load spectra, till thermal analysis on the systems.
It becomes clear that the Workbench tool is used for a wide range of problems taking into account non-linear aspects related to flow, materials, temperature gradients or surface contacts.
According to the conclusions of the virtual prototyping phase, the solutions to be adopted and produced are easily identified. The results achieved by means of the simulations are then compared with those of the experimental tests carried out by the R&D Department, equipped with high pressure test benches, loop circuit for fluid-dynamic measurements, devices for cryogenic and temperature tests and sensors (strain gauges) for deformation measurements.



FEM Analysis of Ship to Fixed Offshore Platform Collision

Collision of a vessel/ship with an offshore platform is a dynamic process involving dynamic factors which could affect the structural response e.g. the way that the collision happens between the vessel and the platform structure, the contact time of collision, the pile-soil-structure interaction during the dynamic response of the platform structure.
For the problem we are speaking about, non-linear responses may take place at some critical regions of the jacket structure, as well as at the impact regions and in the pile-soil interaction.

Therefore, direct integration using explicit method (explicit Ls-Dyna solver) can be assumed as a proper approach to investigate over the ship and platform motion equations and relevant behavior under the ship collision.

There are mainly three design principles considering a ship to an offshore structure collision (see DNV-RP-C204 and NORSOK standard N-004):

- strength design, which implies that the installation is strong enough to resist the collision force with minor deformation, so that the ship is forced to deform and dissipate the major part of the energy;
- ductile design, which implies the opposite, that is the installation undergoes large, plastic deformations and dissipates the major part of the collision energy;
- shared-energy design, which implies that both the installation and ship contribute significantly to the energy dissipation.

The proposed approach can gather the above design principles and give, for each scenario, reliability response both for the ship and for the fixed structure, even if, for what the offshore installation is directly concerned, the ductile design could be the more interesting one once a ship with large kinetic energy has to be considered.

Hence, the impact energy absorbed by an offshore jacket structure impacted from a ship generally involves the following energy absorption processes:

- local denting or crushing of the tubular member section;
- elastic beam bending;

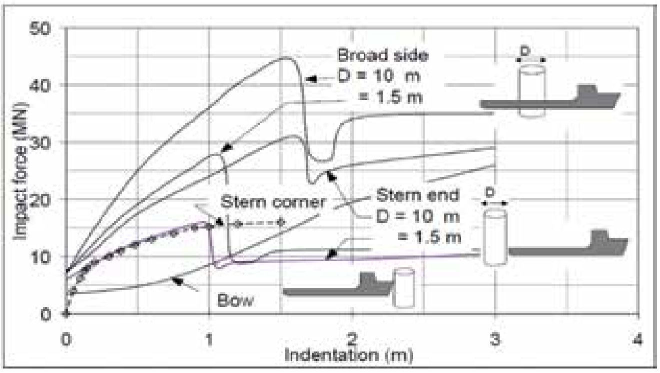


Figure 1 - Force-Deformation relationship

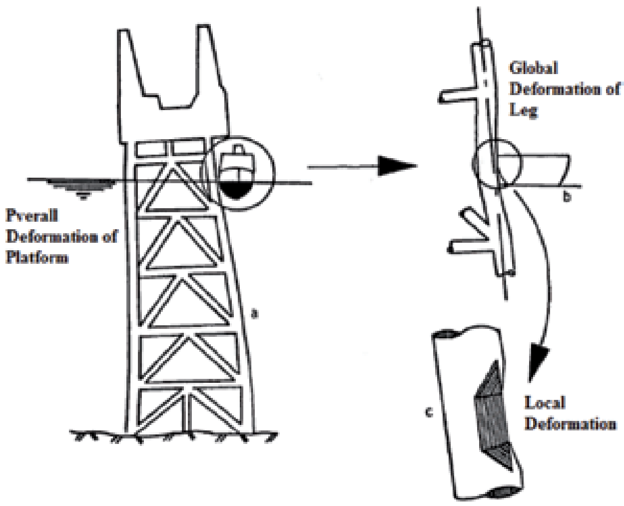


Figure 2 - Deformation modes

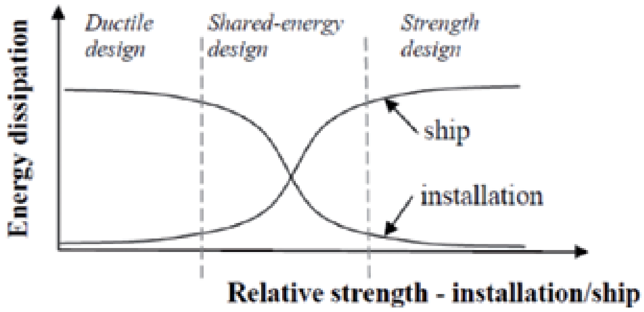


Figure 3 - Design categories

- plastic bending/hinge formation;
- global structural deformation (elastic and plastic).

Therefore a Finite Element Model is used to simulate the ship/vessel (whole or a portion in relation to the complexity and size of its geometry), the whole jacket structure, completed with foundation piles, the top-sides, completed at least with main modules (with their own mass) in relation to a reliable assessing of the offshore installation dynamic behavior.

Vessel

The ship is represented via shell elements, detailing the zones near the point of impact: stern, bow, broadside.
To the ship, proper mass density in relation to the own mass and the added mass have to be assigned.
From the vessel's vertical constraint point of view (the vessel is 'supported' by the buoyancy), we can assume vertical springs located at the vessel wet surfaces and vary with the vessel draft (hence, non-linear springs reacting in relation to the vertical displacement field the vessel assumes during the impact).

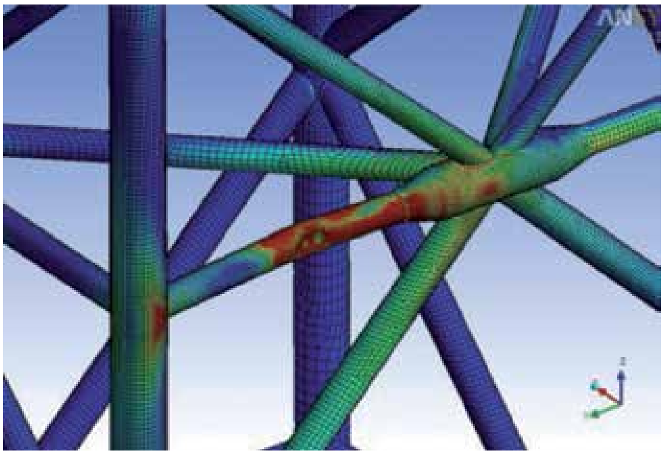
In order to reduce the time consumption, either a prescribed displacement or the design impact velocity, is assigned to a position on the vessel close to the jacket.
To further reduce the computation time, the vessel's motion is restricted to a pre-defined direction, hence insuring that the ship hits the column in a desired way.
Vessel material model provides elasto-plastic behavior with isotropic hardening.

Jacket

The jacket is wholly represented via tubular elements with a formulation able to manage the interaction between axial and bending longitudinal stresses in the plastic domain.
Only at regions where the impact happens, the jacket is simulated with proper level of accuracy, via shell elements (the default shell element type in Ls-Dyna is the Belytschko-Lin-Tsay shell element – this element formulation is used as it is a computational efficient and robust element). Hence, where required, legs, braces, nodes are represented in detail laying the shell elements on the middle surfaces of the simulated tubular members.
According to convergence studies and the need to properly assess the large deformations, high stresses and strains at the jacket columns, shell at dimensions the impact zones are $5 < Le/t < 10$ (where Le is the element length and t is the shell thickness).

In order to properly consider the effect of the foundation behavior on the dynamic behavior of the whole installation, the jacket model is completed with the modeling of foundation piles, represented via beam elements laterally constrained by p-y curves and vertically constrained by t-z curves.

To the jacket model are assigned structural masses related to material density and no-structural masses (like added mass, mass of entrapped water, marine growth, etc.).
Jacket material model provides elasto-plastic behavior with isotropic hardening.



Example - Analysis is used to confirm that the structure can absorb sufficient energy to withstand impact from ship permitted within the safety zone (source: AA Oil and Gas special issue 2014)

Top-sides

In order to adequately represent top-sides stiffness and masses, the idea is to model in a rough way the main structures assigning the masses of the main modules.
Hence, the top-sides model is represented via beam element with linear elastic behavior.

Results

In terms of results the analysis can give: the distribution of stresses and strain on the impacted zone, the extent of the contact areas and of local indentations, the formation of plastic hinges along columns or at the column nodal connections, the cross sectional forces/stresses along columns axis, the force-deformation curves/relationships as function of impact development and time, the ship strength and ship deformation as function of impact development and time, and the energy dissipated by the ship and by the jacket (both due to local plastic and global elastic deformation).

Livio Furlan, EnginSoft

Numerical FE Simulation Applied to Design of Floating Roof for Oil Storage

This article presents a new methodology based on FE analyses applied to the design of floating roof for oil storage. Single pontoon floating roofs have been studied by applying prescriptions of API international standard. Extremely heavy testing conditions have been considered in order to simulate emergency situations which could occur during the roof service life. New approach based on transient FE analysis has been used and a dedicated procedure has been developed. System strength, buckling phenomena and pontoon functionality after the application of testing conditions have been checked and compared for different roof sizes. This work has been carried out in cooperation with Paresa SpA which is a worldwide leading company specialized in engineering, construction and maintenance of large steel structure in particular for Oil&Gas industry.

INTRODUCTION

Floating roofs for oil storage tanks are steel structures typically employed in the oil and gas industry. External floating roof tanks are commonly used to store large quantities of petroleum products such as crude oil. They includes an open-topped cylindrical steel shell equipped with a roof that floats on the surface of the stored liquid. The roof rises and falls with the liquid level in the tank as product is added or withdrawn from the tank. A rim seal system is mounted between the tank shell and roof to reduce liquid evaporation. This type of roofs minimize the vapor space between the pontoon and the liquid surface; since there is no large vapor space for the liquid to evaporate into, vapor losses are also minimized. Main disadvantages of the external floating roof use is that large amount of rain water and/or snow can accumulate on the roof top leading to roof structural problems and, eventually, the roof may sink. Therefore water on the roof must be drained by drain line system. Design requirements for external floating roofs are provided in Appendix C of the API Standard. Limited procedure and rules in design the floating roof are available in the technical literature and these had resulted in lots of floating roof failure during past years. The type of storage tank used for a specified product is mainly determined by safety and environmental requirement, but also operation cost and cost effectiveness are fundamental factors to

PARESA

Fig. 1 – Storage tanks geometry: single pontoon (a) and double deck type (b)

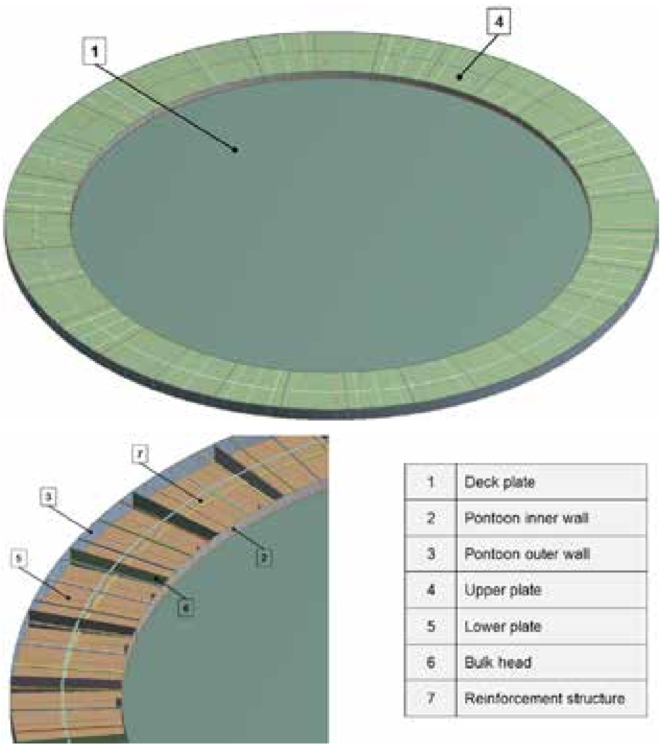


Fig. 2 - Roof modeled geometry and details of main parts

be considered. Design and safety has come to a great concern as reported case of fires and explosion for the storage tank has been increasing over the years causing injuries and fatalities. Spills and tank fires not only generate environment pollution, there would also be severe financial consequences.

As regard the classification of external floating roofs, they can be divided in to two main categories: single deck and double deck pontoon as shown in Fig. 1. In this study single deck type has been considered as its design against sinking is more challenging. The design and construction process is bounded and regulated by various codes and standards:

- American Standards API 650 (Welded Steel Tanks for Oil Storage).
- British Standards BS 2654 (Manufacture of Vertical Storage Tanks with Butt-welded Shells for the Petroleum Industry).
- German Code Din 4119 – Part 1 and 2 (Above Ground Cylindrical Flat Bottomed Storage Tanks of Metallic Materials).

Aim of this article is to present a numerical approach for the detailed evaluation of roof resistance under the action of testing loads according to API international Standard 650. Different sizes has been analyzed to cover the typical range of tanks produced by Paresa SpA. Though the codes addressed the minimum requirement on the pontoon volume, there is no mention on the structural adequacy. No proper procedures or standard rules stated in any code or engineering handbook are available for design floating roofs assessing structural integrity and buoyancy stability. For this reason a numerical methodology based on FE analysis has been developed using ANSYS release 15.0 FE code. As prescribed by the API code, elastic stability against "gross out of plane" buckling and local buckling of the outer pontoon and roof functionality (no significant plastic deformations and tolerable outer rim submergence) must be checked at different testing conditions representing emergency situations, i.e. primary drain inoperative. Two load cases have been then considered:

- LC1: submergence of outer rim when 254 mm rain water is accumulated on the roof (primary drains closed).
- LC2: The submergence of inner and outer rim near the punctured compartments when deck and two adjacent pontoon compartments are punctured.

Both in-situ testing conditions (tank filled with water, nominal thickness of the steel plates) and operating conditions (tank filled with oil, 3 mm corroded thickness of the steel plates in contact with stored product) have been analyzed. FE simulations have been carried out by taking into account large deformation effects and non-linearity of applied loads. For the accurate evaluation of the roof's equilibrium position due to floating effect, a specific transient procedure has been applied for each testing condition to correctly estimate hydrostatic pressure due to submergence level. To evaluate the effect of testing loading conditions on the roofs resistance and functionality, also operating conditions (no accumulated rain water and no compartment puncturing) with stored oil have been simulated.

DECK SIZE		#1	#2	#3	#4
Inside tank diameter	ID [m]	59	43	30.5	21
Tank height	H [m]	17.15	16.2	16.3	16.7
Tank capacity	CAP. [m3]	40000	20000	10000	5000
Roof mass	m_{roof} [kg]	258424	42542	125077	69115
	m_{roof}/Area [kg/m^2]	94.5	122.8	86.1	94.6

Tab. 1 - Summary of analyzed storage tank main dimensions and mass

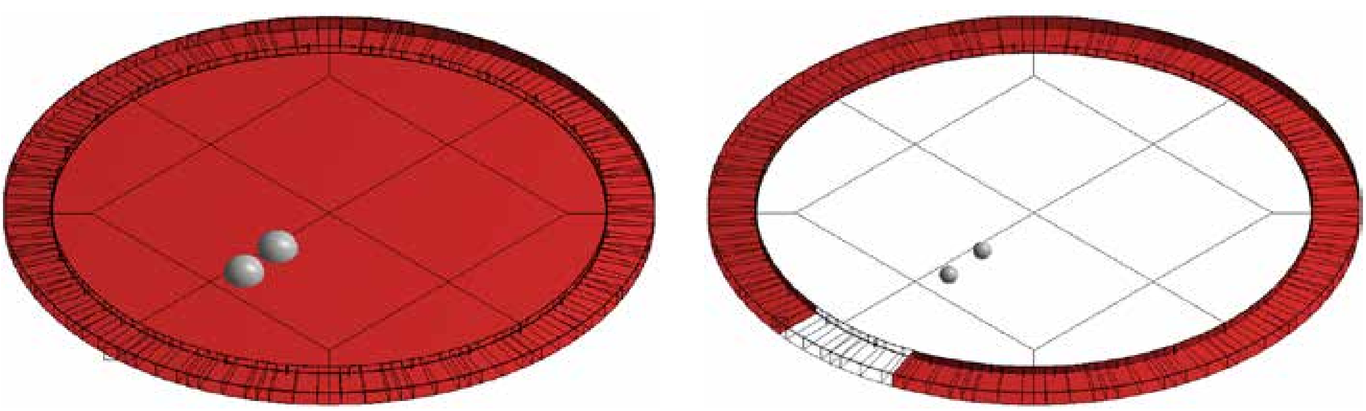


Fig. 3 – Deck lower wet surfaces for LC1 (a) and LC2 (b) testing conditions

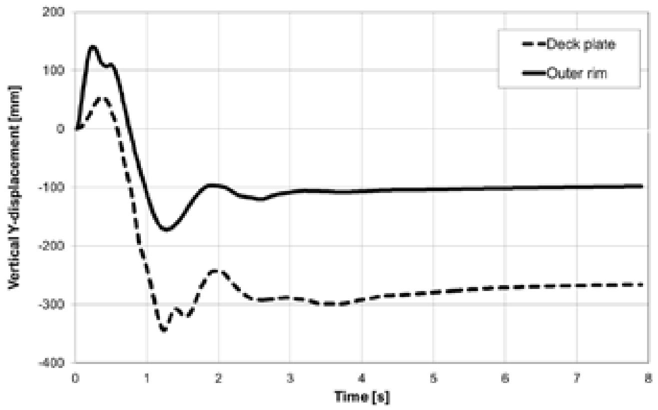


Fig. 4 - Example of monitor point vertical displacement vs. time diagram for transient dynamic analysis

SIZE		#1	#2	#3	#4
Rolling ladder mass	m_1 [kg]	1242	1155	1155	1155
Runway mass	m_2 [kg]	596	944	576	576
Position rolling ladder wheel	R_1 [m]	15.9	2.1	8.9	2.7
Position runway (center of gravity)	R_2 [m]	12.1	5.6	5.4	0.8

Tab. 3 – Mass and centroid position (distance respect to the tank center axis) of rolling ladder and runway

MATERIAL	NOMINAL COMPOSITION	PRODUCT FORM	UTS (MIN) [MPA]	YS (MIN) [MPA]	ELONGTION AT BREAK
SA 283 Gr. C	Carbon steel	Plate	380	205	22%
SA 36	Carbon steel	Bar	400	250	20%

Tab. 2 - Main material properties for deck structure plates and bars

FE MODELS

The study has been carried out on single pontoon roof type with different sizes, with deck diameter ranging from 20 to about 60 meters. The roof total masses and storage tank main dimensions for the different sizes under investigation are listed in Tab. 1. The modeled roof geometry with details of main parts is shown in Fig. 2. Analyses have been carried out by 3D FE model using a combination of 1D beam and 2D shell element for reinforcement structure and deck plates respectively. To simulate the submergence of deck plate due to accumulated rain, volume of water on the deck plate has been considered using proper 3D fluid elements (fluid80) which correctly simulate the fluid behavior and its hydrostatic effect. For all the analyzed roofs a mesh with about 250,000 nodes (1,500,000 dof) has been developed. Deck structure is composed by plates and bars, main material properties for the roof components are listed in Tab. 2. Self-weight of the deck structure and accumulated water for LC1 has been assumed as the main acting load according to the data provided by Paresa SpA. The application of hydrostatic pressure of the stored fluid is described with details in following section. A density scaling has been considered in order to obtain the total deck weight + accessories while the weight of rolling ladder and its runway have been simulated as remote mass applied on the deck plate. Mass and centroid positions of rolling ladder and runway are reported in Tab. 3. Conservatively, the rolling ladder and runway masses have been located in correspondence of punctured compartments for LC2. As can be noticed from Tab. 3, ladder and runway masses are about constant for all the analyzed tanks while roof diameters changes significantly. Therefore, for smaller roof in particular, weight of ladder and runway could critically influence the roof final position and submergence. A FE analysis of the whole roof, as described in this paper, results then fundamental to properly estimate these secondary effects which are difficult to be considered with a classical design-by-formula approach. For LC1 load condition, accumulated water has been simulated explicitly by modeling the water volume on the deck plate and assigning proper 3D fluid elements (fluid80) which can simulate the hydrostatic effect on the deck plate: the hydrostatic pressure due to the differential deck plate submergence which induce a higher water accumulation at the center is considered. Buoyant pressure has been applied on the deck wet surface according to load case under investigation. For LC1, the whole pontoon lower surface has been assumed as wet, while, for LC2 condition, the deck plate surface and the pontoon lower plates in correspondence of the punctured compartments have been considered neutral respect to the buoyancy effect. Conservatively the punctured compartments have been assumed in correspondence of ladder and runway eccentric masses. Wet surface considered for the load cases analyzed are shown in Fig. 3 where also ladder and runway point masses are indicated. According to prescription of API 650, the buoyancy pressure has been calculated by means of the algorithm described in following section assuming water (density = 1000 kg/m3) and oil (density = 700 kg/m3) as stored products respectively for LC1 and LC2 conditions. Standard earth gravity has been applied to the model and no support constraints are needed because the equilibrium is calculated step by step with the transient analysis procedure according to buoyancy force. For LC1 condition, a remote point has been assumed on the top surface of the accumulated rain water volume to impose the free surface condition for fluid elements. No vertical displacement constraint has been imposed to this remote point because the accumulated rain water free surface can change during simulation according to the deck plate deformation maintaining the total water volume. To prevent rigid body drifting in horizontal directions, and yaw rotation about vertical axis of the whole roof, tangential displacements of 4 control points at 90° on the outer rim plate have been constrained.

ANALYSIS PROCEDURE

Because of the roof is a floating structure, its equilibrium position assumed due to buoyancy effect is not a priori known and it depends on the whole roof self-weight and vertical deformation. Therefore, the hydrostatic pressure acting on each point of the roof wet surface is dependent on the vertical

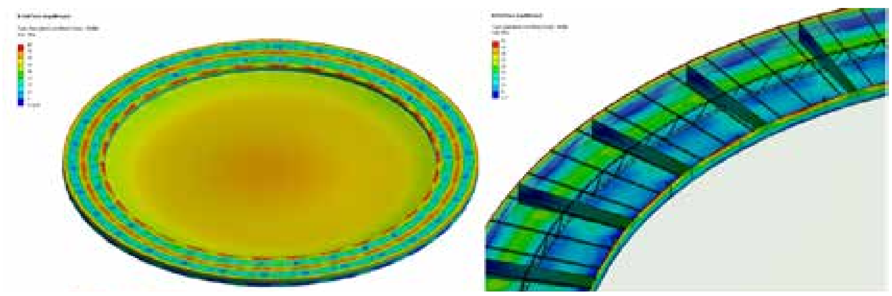


Fig. 5 – Example of von Mises stress for deck #1 and detail of stress on internal compartment plates

displacement of structure. For this reason, a transient procedure has been developed to simulate buoyancy of the floating roof. ANSYS solver can apply a spatial pressure gradient with various techniques, including the SFFUN command which uses for input an array containing hydrostatic pressures at surface node numbers calculated from node depth values. These pressures must be updated during SOLVE steps as a function of depth when a structure moves vertically in a liquid's hydrostatic pressure "field" in large displacement analysis. A dedicated procedure has been developed and main stages of the calculation algorithm are reported below.

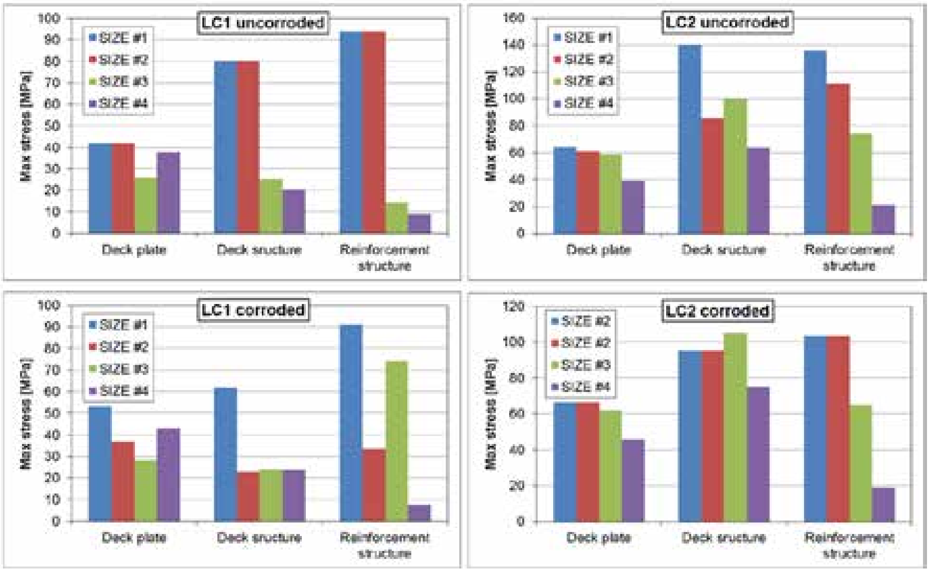


Fig. 6 – Summary of max equivalent stress for different analyzed decks

STAGE 1. Mass and Volume calculation

A preliminary small displacement static step has been solved to precisely calculate the total mass of the structure and its volume based on the meshed elements. For this step solid elements filling the pontoon compartments with air (density = 1.2 kg/m^3) has been used. The purpose of the displacement volume is to calculate a reasonable mass damping value for the transient analysis.

STAGE 2. Transient analysis

Full transient dynamic large displacements step has been then solved to calculate structure equilibrium position due to buoyancy. The hydrostatic pressure has been placed at exterior nodes on wet surface as shown in Fig. 3 for LC1 and LC2 conditions by considering a stored liquid free surface at Y=0, where Y is the vertical direction. Buoyant force has been calculated from the product of displacement volume and a liquid density. Nodes on wet surface under the liquid free surface are subjected to a pressure proportional to depth, while for nodes above free surface the pressure is set to zero. The final time step has been assumed long enough to reach the equilibrium: monitor points have been used to control the vertical deformation of the structure (plate and pontoon) and transient simulation stops when a plateau is reached in the displacement vs. time diagram. In order to initialize the transient calculation whose final result is the effective deck position, deck has been assumed to be already under the water level at the initial stage. The more the starting position is close to the equilibrium submergence level, in fact, the more the total time for the equilibrium is short and secondary effects such as local oscillation of the deck plate can be reduced. To avoid bobbing instability of the structure during the transient calculation step, a mass damping has been applied to the system with ALPHAD command. The calculation of damping factor has been done by estimating a characteristic stiffness for submersion considering cube root of structure volume as a characteristic length and liquid weight density (kg/m^3 * 1g) as a buoyant force. Given the mass of the structure and a characteristic equivalent stiffness for the buoyant force, a critical damping factor for a 1 degree-of-freedom damped system has been estimated. About 70% of this critical damping divided by total mas has been used as argument of ALPHAD command. For the total time period of the transient analysis, a characteristic time for a bobbing cycle has been calculated by considering a 1 degree-of-freedom undamped free oscillation. Due to the high size and mass of the floating roofs and its low global bending stiffness, a total simulation time equal to 2 bobbing cycle is enough for the equilibrium position calculation. Time integration is performed by executing a series of SOLVE commands inside a loop. After each integration step, the hydrostatic pressures are updated with new vertical position for each node on the wet surface. Time steps have to be small enough to maintain accuracy and avoid divergence. Number of time steps ranging from 200 to 600 has been used in the simulations for different roof sizes. Higher number of steps when vertical velocity of structure is high, i.e. higher mass to volume ratio, initial position far from the final one which induce high vertical position variation at the beginning stages and/or high structure deformation during the transient cycle. Example of plot of vertical Y-displacement diagram for size #1 roof taken during transient analysis shown in Fig. 4 point out that less than two bobbing cycle have been necessary to reach the equilibrium position.

ASSESSMENT

According to API standard, both resistance and functionality criteria have been assumed. Acceptable stress level, i.e. maximum stress under the allowable limit, elastic stability against "gross out of plane" buckling and local buckling of the outer pontoon and maximum submergence of the outer rim have been checked for all the load condition analyzed. According to API STANDARD 620 allowable stress of 154 MPa and 188 MPa have been assumed respectively for plates of deck structure and for beams of reinforcing frame. According with API standard 650 section C.3.4.2, the pontoon shall be designed to have adequate strength to prevent permanent distortions and the allowable stress and stability criteria shall be jointly established by the Purchaser and the Manufacturer. Therefore, being LC1 and LC2 not standard loading operating conditions, API standard 650 does not define a mandatory criteria for allowable stress. With conservative assumptions the allowable stress in the floating roof has been taken equal to 3/4 of YS both for testing and design condition. The value is considered reasonable in assessment of testing conditions. With such assumption, the required level of safety is the same of the shell plates in hydro-testing condition (ref to API 650 section 5.6.2.2).

Fillet welds have been also checked according to the same standard by considering specific joint efficiency (safety factor on the allowable stress) of 0.7 for double lap joints on deck lower plates and 0.35 for single lap joint on deck upper plates. In compliance with API functionality acceptance

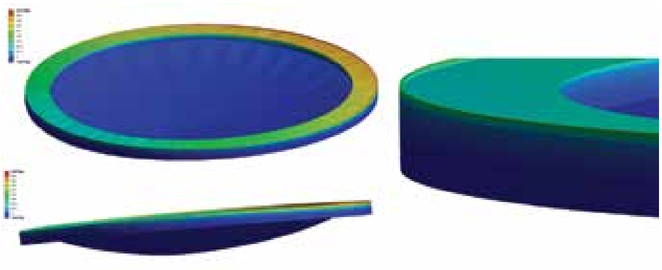


Fig. 7 – Example of submergence for roof size #3. Areas under fluid level are represented in blue

criterion, maximum submergences within 65% or 95% of the outer rim height in in testing and operating condition respectively are allowed. As regards the elastic stability acceptance criterion, the nonlinear transient analyses with large deformation option which have been performed allow the evaluation of the equilibrium condition for deformed structure and can guarantee that buckling of pontoon element does not occur for the load condition considered (buckling effects are implicitly monitored by the analysis that would result in non-convergence of the solution if buckling occur under actual loads). To evaluate gross out of plane buckling and local buckling of the outer pontoon for further load levels linear buckling analyses starting from the equilibrium condition calculated by means of transient analysis have been carried out. Therefore load factor for the first buckling mode shape must be higher than 1.

RESULTS AND DISCUSSION

In Fig. 5 example of equivalent von Mises stress map on deformed shape for deck size #1 with detail of the internal compartments plates is shown. For all sizes deck plates are mainly stressed at its central area due to accumulated rain water as expected. As regards outer pontoon, most critical areas are located in correspondence of intersection between walls and upper/lower plates. Stress intensification also occurs at junction between reinforcement structure and upper/lower plates. Stress assessment criteria for roof sheets, reinforcement structure and welds are satisfied for all the analyzed roof. Maximum stress found for different analyzed structures are summarized in Fig. 6. Results pointed out that stress in deck components are generally higher for sizes #1 and #2. For larger decks in fact, due to the lower stiffness and the considerable load induced by the accumulated rain water and self-weight of deck plate, higher bending deformations and stress occurred in particular at the upper plate and in correspondence of inner and outer rim joint. Results pointed out that testing condition generates severe stress status respect to normal operating conditions particularly for larger decks. Fig. 7 shows an example of submergence map for roof size #3 at LC2. Areas under the fluid free surface are represented in blue. Detail of outer rim submergence near the rim top limit where the gasket is mounted is also shown. In Fig. 8 the submergence levels at all analyzed conditions for different roof sizing are summarized. Results pointed out that assessment criteria are always satisfied. Both analyzed testing conditions proved to be severe from the point of view of roof submergence, confirming that a detailed FE analysis is needed to properly evaluate the roof functionality during in situ tests. Submergence levels for small roofs resulted critical respect to largest ones. The effect of accumulated rain water, reduction of buoyant surface and the influence of ladder and runway masses, in fact, have a major impact for roofs with lower diameter. Elastic stability checks are summarized in Fig. 9, where minimum load factors are reported at each analyzed condition.

Buckling load factor calculated are always higher than the limit even if, especially for larger roofs, values near 1 have been found. Larger pontoon in fact are more subjected to local buckling phenomena due to the lower stiffness of their components. However, for all roof sizes, first modes are related to local buckling of bulk head plates or struts of reinforcement internal structure. Gross out of plane buckling have been found only for high load factors (> 10) pointing out that no risk of roof sinking due to structure bucking is present. A dedicated methodology for the design and analysis of floating roofs for oil storage tanks has been developed and applied to a range of roof size. Extremely severe testing conditions have been assessed in compliance with widely applied technical standard. The approach proved to be robust and allow to analyze with details several operating situations usually not systematically considered in the design stage. Therefore, a systematic application of the proposed procedure in the design chain for this kind of structure should be considered to improve the confidence level about roof behavior in testing and operating conditions and reduce risk of pontoon failures. Results pointed out that, generally, testing stages are very severe from the point of view of induced stress, submergence level and buckling risk respect to normal operating conditions. For larger pontoon in particular, stress levels and buckling phenomena could be assumed as driving factors during the design, while, for smaller pontoon, submergence level should be carefully checked especially in case of compartment puncturing.

Stefano Cavalleri, Sergio Sarti - EnginSoft
Davide Montalti - Paresa

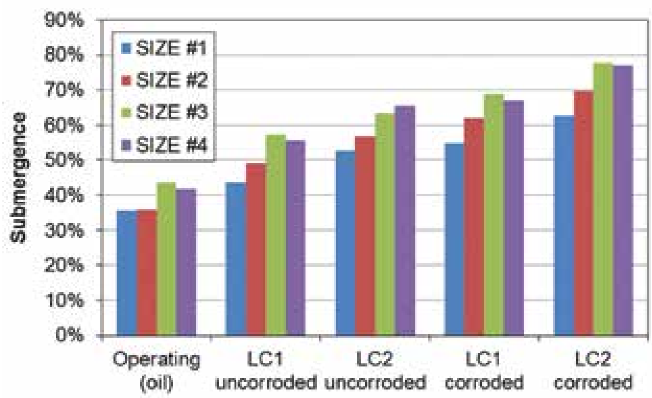


Fig. 8 – Summary and comparison of submergence levels

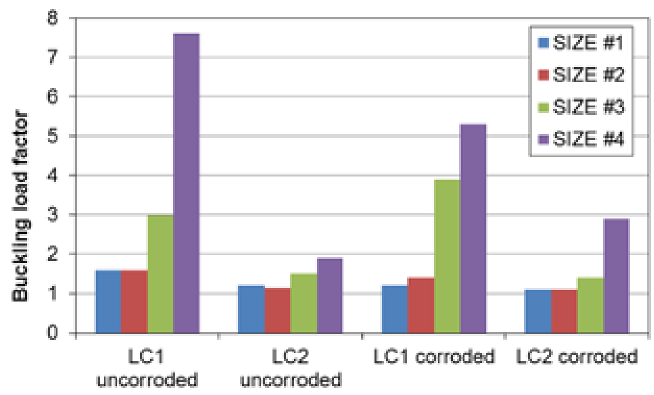


Fig. 9 – Summary of minimum buckling load factors

FSO and Shuttle Tanker in Tandem Configuration Hydrodynamic Analysis Finalized to the Structural Verification of the FSO Mooring System

Strength and Fatigue Verifications of an FSO mooring system have been performed basing the results on proper hydrodynamic analysis (developed inside ANSYS-AQWA) and structural analyses (developed inside ANSYS-Workbench) of the system and relevant components.

Hs (m)	Tz (s)
1.5	4.1
2.0	4.7
3.0	5.7

Sea-state assumed for the analysis in offloading condition

Hydrodynamic Analysis

The FSO (109.000 DWT), operated by Edison, is moored on the Rospo Mare Offshore Oil Field. The FSO mooring is guarantees via 6 chains connected to a rotating turret, installed at the FSO bow. During the oil offloading operation, the Shuttle Tanker (45.000 DWT) is moored, via an hawser, at the FSO aft end.

The offloading operation takes place under proper sea conditions, with waves characterized by significant height (Hs) ad zero up-crossing period (Tz). To each sea state, consistent current and wind have been accounted for.

The hydrodynamic model (performed inside ANSYS-AQWA suite), simulating the FSO and the Shuttle Tanker (this one moored, at its stern, to a Tug via a mooring cable), refers both to aligned and misaligned meteo conditions (current incoming at 50 degrees with respect to wave direction, wind incoming at 25 degrees with respect to wave direction).

On the model (FSO + Shuttle Tanker + mooring lines), time domain hydrodynamic analysis has been performed for each defined sea-state, obtaining, for each mooring chain and for the hawser connecting FSO and Shuttle Tanker, the axial tension as function of time.

In order to check the strength resistance of mooring components (such as Chain Stoppers and 'Ecubier') installed at the rotating turret, besides hydrodynamic analyses under offloading conditions, also hydrodynamic analyses of FSO in moored condition, for extreme storm case (100 years return period), have been performed.

Strength and Fatigue Verification of Chain-Stopper and "Ecubier"

Based on results of hydrodynamic analysis performed for both extreme and offloading conditions, strength and fatigue verifications of Chain Stopper and 'Ecubier' have been performed.

Strength checks have been based on results obtained from contact non-linear analysis performed of Finite Element Model of Ecubier + Chain Stopper under extreme load case (practically the chain minimum breaking load).

Fatigue checks have been developed according to spectral approach as required by DNV OS-E301 (Position Mooring), assuming proper S/N curve data as reported in DNV RP-C203 (Fatigue Design of Offshore Steel Structures).

The assumed hypothesis at the base of fatigue spectral approach is that the stress range, S, is a random variable characterized by a probability density equal to p(S) and that, for each sea-state, the number of cycles having stress variation in the range of S and S+dS is directly related to ni p(S), where ni is the total number of cycles of that sea-state.

Fig. 1 - Hydrodinamic Model of FSO, Mooring Lines, Shutter Tanker

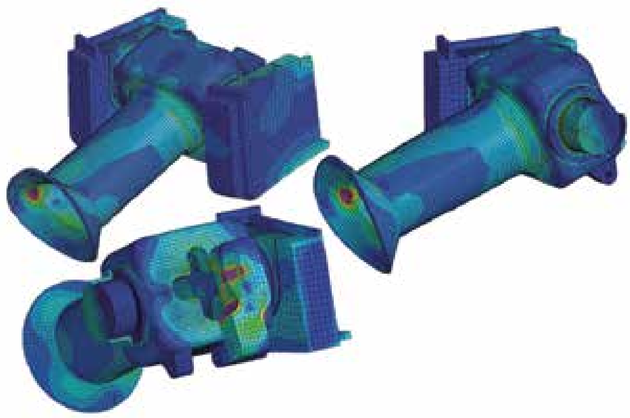


Fig. 2 - Von Mises Stress distribution on Ecubier and Chain Stopper

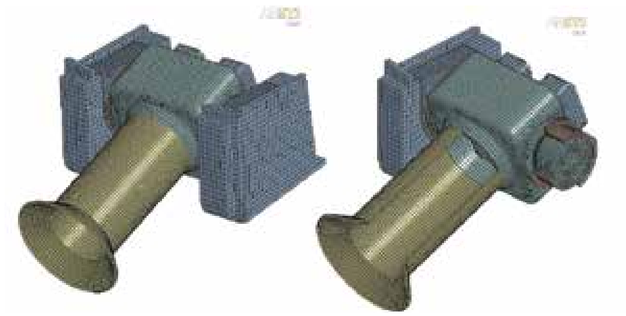


Fig. 3 - Finite Element Model of Ecubier and Chain Stopper

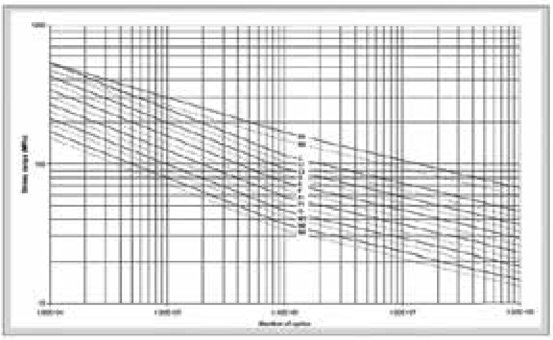


Fig. 4 - S/N Curves in sea-water with cathodic protection

Based on this and on the fact that, for offshore structures, the probability density of stress ranges, p(S), can adequately be represented by a Rayleigh distribution, the damage, Di, for the i sea-state, is given by the following relation:

$$D_i = \frac{n_i}{\bar{a}}\left[\left(2\sqrt{2}\sigma_s\right)^m (m/2)!\right]$$

where a and m are factors of S/N curve (C curve has been considered for fatigue verification of Ecubier and Chain Stopper), while σs is the standard deviation of S distribution.
Finally, based on Miner-Palmgreen relation, the total damage, D, due to the summation of damages of each sea-state, Di, is:

$$D = \sum_i D_i$$

Enrico Miorin, Fabiano Maggio, Livio Furlan
EnginSoft

Design and FEM Analyses in Offshore and Oil&Gas Industry

Besides competencies in Automotive, Aerospace and Industrial Engineering Simulations, EnginSoft has knowledge also in the Design and Analyses voted to the Oil&Gas and Offshore Industry. Many consultancy activities have been performed via collaborations with the most important Italian players in this sector: ENI, Saipem, Tecnomare, MIB Italiana, Petrolvaves, Cameron, FBM, Officine Resta, Nuovo Pignone, ATB, Foster Wheeler.
EnginSoft can supply a full range of services covering projects entire design route, from the earliest conceptual studies passing through FEED and basic design up to detailed design and installation engineering.
The following list reports some of the Oil&Gas Business Unit competences:

- Conceptual and detailed design and structural analysis of fixed offshore platforms (jacket, top-sides, buoyancy tanks, stiffened structures)
- Design and analysis of subsea foundation templates
- Design and analysis of pressure vessels, valves, piping, rack, etc.
- Design and analysis of subsea manifold (even for installation, repairing and retrieval operations)
- Detailed structural analysis of structural parts (Hulls, Deck, etc.) of Semi-Submersible Vessels
- Detailed structural assessment of steel Gravity Based Structures (GBS) including stiffened plate code checks
- Detailed design and structural analysis of risers and FPSO's mooring connectors
- Revamping of fixed offshore platforms (assessment of structural reliability- re-certification and life extension), fracture and fatigue assessment of installed jacket structures (risk analysis)
- Motion Analysis of Floating Vessels (even for Marine Pipeline Installations)

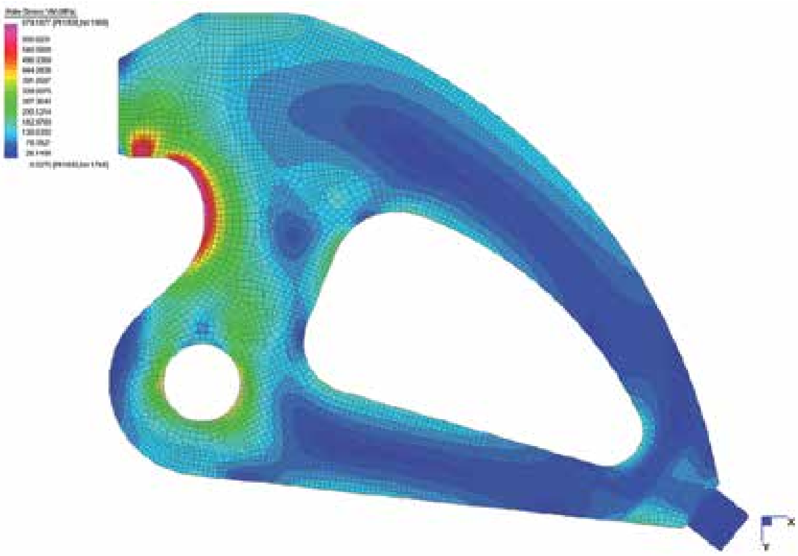
The BU, which is located in EnginSoft Padova Office and is coordinated by Livio Furlan, has high skills also in the field of structural and mechanical applications in general (as an example the design and analysis of Roller Coaster structures and cars or the design of large valves for hydroelectric power plants).

Livio Furlan, EnginSoft

QCDC connector design on a FPSO unit

In the off-shore oil and gas industry, in deepwater locations or in remote field locations fixed off-shore processing facilities and seabed pipelines to connect to an onshore terminal are no longer cost effective. Instead in these cases the so called Floating Production Storage and Offloading (FPSO) vessels are employed. FPSOs are floating vessels and can be either a conversion of an oil tanker, or a vessel built specifically for the application. They are often used in small oil fields as well, which can be exhausted in a few years and do not justify expensive structures and pipe systems.

A FPSO (which is often referred to as floating production facility) has various systems for handling and separating the different hydrocarbons, as well as mooring systems and system for the dynamic positioning. These have to be designed against severe sea states, which can occur in the operational life of the unit and of its components. In a word, a FPSO must be operatively secured in all the weather and sea design conditions.



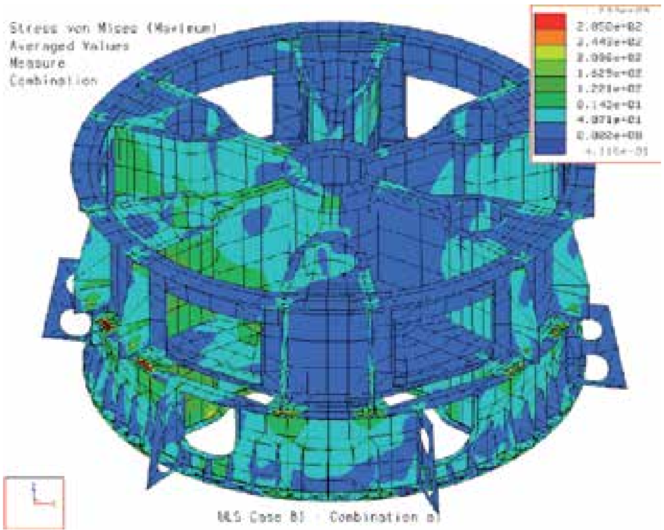
Clamp Stress

Specifically, should the weather and sea condition exceed the design operating conditions, the vessel has to be free to leave the site in order to avoid possible damages to the structures and to the pressure pipelines that, by means of the well-heads located on the seabed, transfer the hydrocarbons from the oil pool to the FPSO. Similarly, when the oil field is exhausted, the FPSO has to be disconnected to be relocated in a new oilfield.

Therefore it is necessary for the FPSO to be equipped with a Disconnetable Transfer System (DTS). DTS are used both on FPSOs anchored to the seabed by moorings or on FPSOs with dynamic positioning. In the former case, the DTS is at the same time the mooring system of the vessel and the connecting systems of the risers coming from the submarine well-heads.

One of the main components of a DTS is the Multibore Quick Connector Disconnector Coupler (QCDC). This is the very heart of the connection of both the hydrocarbon transfer lines (risers – that is pressure pipelines – and umbelicals) and mooring system. The QCDC is a device formed by two halves. The upper one is located in the FPSO moon-pool, whereas the lower one, also connected with the riser-buoy, is linked, under operating conditions, to the upper one by means of a clamping mechanism mounted on a rotating collar operated by a couple of hydraulic cylinders. The riser-buoy, in turn, is a bouy gathering risers, umbelicals and moorings and allowing for the recovery of the lower part of the connector once the bad weather conditions are over and the unit can get back to the operation conditions.

Each production line, and each water injection line, on both sides of the connect are equipped with valves. In case of programmed or emergency disconnections, these valves close just before the clamping system is activated and the lower part of the connector



QC/DC Structure – Von Mises stress for ULS B1-a Case

is released. Disconnection is normally a controlled process and it includes the risers washing and the riser-bouy lowering, before the units leaves the site.

Designing a multibore QCDC is a complex engineering task involving advanced knowledge in designing pressure vessels as well as structural systems. Just to mention some typical figures, riser's pressures are in the range of 520 bar, and resulting buoy 'axial' load is in the range of 20000 KN. Moreover a variety of design standards have to be taken into consideration, since they apply to the different components of the system.
In this context EnginSoft boasts a long-term well-established collaboration with MIB Italiana, a world leader supplier of multibore QCDC.

Here reference is made to the example of a QCDC with:

- six 6'' lines of production and injection;
- three umbelicals (Hydro/Electric Stab Plate Housing);
- three housings for dampers which are active along the re-connection phase between the lower and the upper halves of the connector.

The connector has been designed against normal operating conditions, extreme operating conditions, offshore pressure test condition, and hydrostatic test condition. A fatigue analysis has been performed as well, to evaluate the impact of the variations of the axial load transferred on by the risers and by the mooring lines.

Design standards include:

- DNV-OS-E201 – Oil and Gas Processing Systems;
- DNV OS-F201 – Dynamic Risers

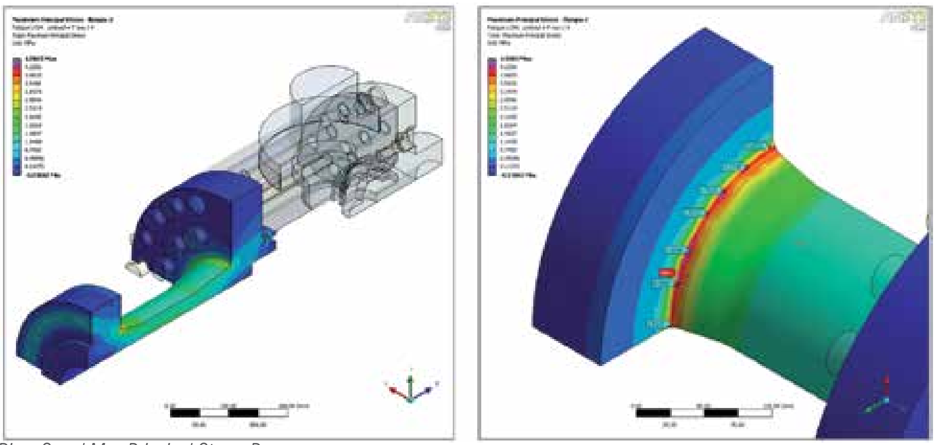
Recommendations/restrictions suggested by the following standards were considered as well:

- ASME BPV Code Section VIII – for pressure vessels;
- ANSI/API Specification 6 – for bolted joints in pressurized components;
- DNV-OS-C101 – Design of Offshore Steel Structures – for structural items;
- DNV-RP-C203 – Fatigue Design of Offshore Steel Structures.

FEM models have been largely used, and specifically shell-type models for the structural components (upper and lower part of the connector, upper spool connecting the QCDS to the rotary table, and lower spool connecting the QCDC to the riser buoy) and fully 3/D models for pressure components.
It has to be stressed that this type of connector is unique of its kind. Trial- and-error procedures do not apply. Efficiency and risk have to be assessed upfront, during the design phase. Shop tests are applicable at the end, but they are just a mean to confirm that the design was correct. That is: the simulation based approach is the only one which can efficiently drive to the correct sizing of the structure and its components, as well as to evaluate different what-if scenarios, delivering the required robustness.

Livio Furlan, EnginSoft

Disconnectable Transfer System



Riser Spool Max Principal Stress Range

Interview to Eng. Paolo Monti
Saipem SpA

Saipem is one of the global leaders in the Engineering & Construction and Drilling businesses, with a strong bias towards oil & gas-related activities in remote areas and deep-waters. Saipem excels in the provisioning of engineering, procurement, project management and construction services, with distinctive skills and capabilities in the design and execution of large-scale offshore and onshore projects. The Company also offer cutting-edge technologies for the gas monetization and heavy oil exploitation.

Inside the Offshore Business Unit, since 25 years Eng. Paolo Monti is involved in Computer Aided Engineering (CAE) activities. CAE experience involves the design and the installation of deepwater pipelines, the development of innovative technologies for the exploitation of hydrocarbon reservoirs in deep - ultradeep waters, the use of non-conventional/innovative materials for offshore oil&gas applications, the design and the maintenance of pipelaying vessels (including topics like structural verification, naval and hydrodynamic studies, simulation of Dynamic Positioning Systems).

1. Which place does (or should) innovation cover in industry/business?

The future of Italian industry can't disregard innovation if it wants to have its international competitiveness granted in the time, with respect to those emerging countries able to provide products and technologies consolidate at lower costs.

Therefore in my opinion innovation and development of new technologies must gain a larger and larger space, not only inside big industrial companies, as already it happens, but also in smaller organizations in a more widespread way.

This fact, which in my opinion will become soon a more and more clear requirement, can't abstract from a cultural evolution, which allows to contend with the market taking a new attitude, leading to walk through technologically different and innovative paths.

2. Which strategies help to be innovative and which evaluations lead to innovation?

In my opinion there are three kind of strategies to be innovative. First of all, as told before, a cultural evolution is necessary, starting to look at innovation not as a problem or a risk but as a resource and a vehicle for competitiveness.

The second strategy is to identify in advance those technologies and those innovative products that the market will demand in the future.

Finally it is important to have the knowledge inside, together with the competencies and the tools (CAE included), necessary to manage and develop innovation: it could be in fact dangerous to delegate or give in charge to someone out of the company what is not well known inside.

3. Which is the role played in this direction by CAE tools and by virtual prototyping?

The capacity of virtual prototyping and the performances of computational tools have reached so high levels to be able to reproduce reality faithfully with reduced costs. This permits to optimize the component design, to improve performances and reliability and to reduce at minimum the experimental tests, even referring to accidental events and/or extreme behavior to which the component could be exposed.

For the design and installation of pipes at 1500-2000m under the sea it is necessary to develop complex numerical simulations, both thermic and structural, in order to guarantee their structural reliability during their entire lifetime, considering that a little damage could have important environmental and economic consequences. The numerical simulation is fundamental even to forecast the movement of naval transportations as pipe-ships and drilling ships, which operate in more and more hostile environments.

4. How have the requests of users changed in the last years?

The demands of CAE and virtual prototyping tools seem more oriented to simulate complex geometries than to study complex, nor conventional or extreme behaviors.

However I believe that the way to technological innovation should be gone through by working for the development of more complex, dynamic and/or nonlinear analysis, above all to improve the components' design respect to extreme behavior and/or accidental events.

This is the specific request of numerical simulation supporting the design and the installation of oil system undersea. The structural components are simple from a geometric point of view; however the study of operative difficulties, to which they are subjected, asks for tools able to develop nonlinear or complex analysis.

5. Which advantages have you found during your professional experience and how has your approach to design/production changed?

Numerical simulation has always been a fundamental element for offshore engineering. In the past it found its place mainly in the study of non-conventional problems, accidental events or in the characterization of the boundary sate of structural components. Today it has become a determining component in the entire engineering process, having the scenarios become more critic and challenging.

For example in the past the design and installation of undersea pipes in not very deep waters could be developed with simple structural computations; today instead the design and installation of undersea pipes in deep water can't ignore a complex and nonlinear calculation, which needs robust codes from a numerical point of view.

6. Which is EnginSoft contribution and how has EnginSoft been able to value quality, potentialities and capacities of your company?

EnginSoft provides an effective support to Italian industry, offering knowledge and tools necessary to numerical simulation, as even supporting the same industry when it decides not to have an inner competence on simulation.

However our relationship with EnginSoft is limited at the moment, as we have our competences and tools for simulations development necessary for the many activities of offshore engineering.

7. Which prospects do you see for the calculation codes in the future challenges?

The codes' producers seem to support the requests of CAE and virtual prototyping tools' users, which are more oriented to simulation of complex geometries than the study of complex, nonconventional or extreme behaviors. I then believe that the way to the development of more complex analysis must be walked not only by industry but even by the codes' producers.

8. Which projects, objectives and new goals do you intend to gain using these tools?

The future goals and aims for numerical simulation are driven by the evolution of oil systems in the sea, especially by the necessity to characterize better and better the structural reliability of systems and components, asked to operate in more and more difficult conditions, deeper and deeper and/or in more and more hostile environments from a meteorological point of view. The main goal is to guarantee a high safety level through numerical simulation, considered the large human, economic and environmental impact in case of failure or damage of the system or of a single component.

9. What do you wish for scientific technology which always look for an ideal dimension between creativity and competitiveness?

The world of scientific technology, university included, must aim for giving a high competence not only executive but even theoretical and technological to Italian industry through the right training for graduates and DPH, in order to help the same industry to have competencies and knowledges necessary to keep the capacity of innovation, which is guarantee of international competitiveness in the time.

Reliability Analysis of Offshore Structures and Inspection Planning

The need for risk assessment and reliability criteria is becoming the major requirement for offshore structures, where probabilistic analyses are achieving progressively greater importance.

EnginSoft, always committed in research and training, is collaborating with Padua University on the reliability analysis of offshore structures, including the support of a PhD. grant within the Industrial Engineering Department.
Typical offshore platform components include "steel jackets" (those parts of the structure immersed in water) and "topside components" (those parts above water). Within this assembly, the most critical elements are the welded tubular connections.
The main objective of the research project lies in the development of routines to assess structural reliability based on unambiguous and scientifically-recognised criteria, with a view to the accurate quantification of operational life and the planning of appropriate inspection protocols.

The aforementioned issues deal with probabilistic structural design field, where both loads (dependent on wave and wind forces) and geometrical parameters and materials resistances are described in terms of stochastic quantities. To cope with this type of problems it is necessary to make use of specific regulations which describe the random nature of the problem. These standards(based on the LRFD approach described below) have been developed according to a probabilistic logic and provide a valuable tool for the verification of the structural components, ensuring predetermined reliability levels. Within the research project, the concept of the reliability of a single component has been reviewed and the process further developed to address systems level performance. The principal Operations Research strategies have then been implemented to identify the dominant failure modes of the structure. Finally, these issues have been reviewed in the context of inspection planning, with the ultimate aim of increasing system safety and reducing the costs connected with offshore structures construction and maintenance.

WSD and LRFD Standards

The design approach of the structures in general and those which, in particular, work and live in the offshore environment, is governed by two distinct design philosophies, identifiable in WSD approach (Working Stress Design) and LRFD approach (Loading and Resistance Factor Design).

The first methodology considers the combined effect of the forces acting on the component or on the structure, reducing, by a factor of safety Ω, the ultimate resistance Rn of the member in such a way that the state of resulting effort is comparable with a reference limit (allowable stress) below the yield point of the material. Although this is a relatively simple method, there is no universal method for determining the coefficient Ω, which is mainly based on past experience and on the need to remain 'far-away' from the materials performance limits.

Otherwise, the LRFD approach provides a more rational and rigorous design alternative. According to this methodology, the overall effect of the load combination acting on a member is determined by amplifying the characteristic value of each load with a load factor γS, that depends on the load type and the likelihood that more loads are present simultaneously. Thus, the overall effect is maintained below a certain reference value, related to the ratio between the nominal resistance (characteristic value) Rn and the resistance factor γR (see Fig.1).

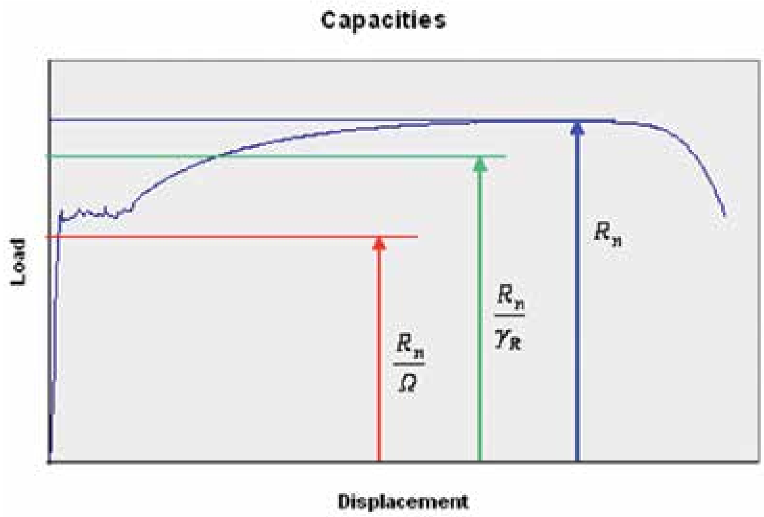


Fig. 1 - Difference between WSD and LRFD standards

Through the introduction of the load and resistance factors, the uncertainties resulting from applied loads and those related to the resistance characteristics of the member are treated separately. These coefficients are determined in a semi-probabilistic way, through a procedure known as "design code calibration" (or also "calibration of partial safety factor").

Conversely, the WSD approach is largely dominated by the deterministic philosophy, whose intrinsic characteristic is the presence of high safety factors, with consequent oversizing and high associated costs of realization. Moreover, the determination of Ω based on the behavior of existing structures can lead to the adoption of inappropriate or even obsolete solutions for new generation structures, thus representing a potential source of danger.

Hence the development of a branch of civil engineering based on probabilistic codes, known as "structural reliability analysis". It is particularly important to develop this approach for structures which operate in highly random environments, and especially so where their failure might have serious repercussions for security, cost and the environment. Offshore structures are therefore exemplary candidates for such an analysis approach.

Component Reliability Analysis

Ensuring the high performance of an offshore structure requires its verification for safety and serviceability with respect to limit state criteria. The term "serviceability" refers to all those issues (vibration, deformation, etc.) that reduce system performance but do not represent a security threat. The term "safety" refers to the absence of collapse or damage in the structure. Safety is ensured by applying design criteria defined by limit state equations for ultimate failure and fatigue.

Introducing an Ultimate Limit State function, g(X), as the difference between the resistance R of the member and the stress (or loads combination) S acting on the member (in other words, g(X) = R-S), it is possible to see that the component is safe when g(X) > 0. Thus, the probability of failure Pf is equal to the area subtended by the distribution of g(X) for values less than zero (see Fig.2). Given the joint probability density function fR,S(r,s), the probability of failure Pf is defined by the following expression:

$$P_f = 1 - \mathcal{R} = P[g(R,S) \leq 0] = \int_{g \leq 0} f_{R,S}(r,s)\, dr ds$$

Otherwise, the reliability R is defined as the probability of the resistance being greater than the applied stress. In the hypothesis of independence between R and S the following well-known formula can be derived:

$$\mathcal{R} = P(R \geq S) = \int_0^{\infty} f_R(\sigma) \left[\int_0^{\sigma} f_S(s) ds \right] d\sigma$$

The difficulties associated with the calculation of Pf are numerous. Generally, the limit function g(X) is a nonlinear function of X and consequently the domain of integration g(X)<0 is not linear either. Moreover, in real problems, the number of random variables involved is high, so that a multidimensional integration is required, which may have a high computational cost. To overcome this issue numerous methods have been developed for the approximate calculation of Pf. Of these, the "First Order Reliability Method" (FORM) is the most frequently used.

The FORM method operates in the U-space of uncorrelated standard normal variables, obtained through the Rosenblatt transformation of the original X-space, as illustrated in Fig.3. The reliability is then calculated by linearizing the limit state curve in the neighborhood of the most probable point (MPP, see Fig.4) and using the following analytical solution for linear limit functions:

$R = \Phi(\beta)$

where β is the reliability index, defined as the distance of the MPP from the U-space origin, i.e. the minimum distance of the curve g(U)=0 from the origin. Consequently, the calculation of R is reduced to a minimum optimization problem (HLFR algorithm, see Fig.5).

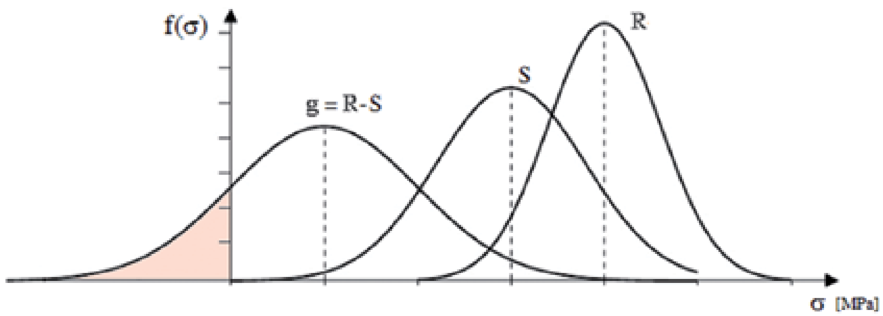


Fig. 2 - Probability density functions relative to material resistance, stress in the member and their difference, i.e. the ultimate limit state function

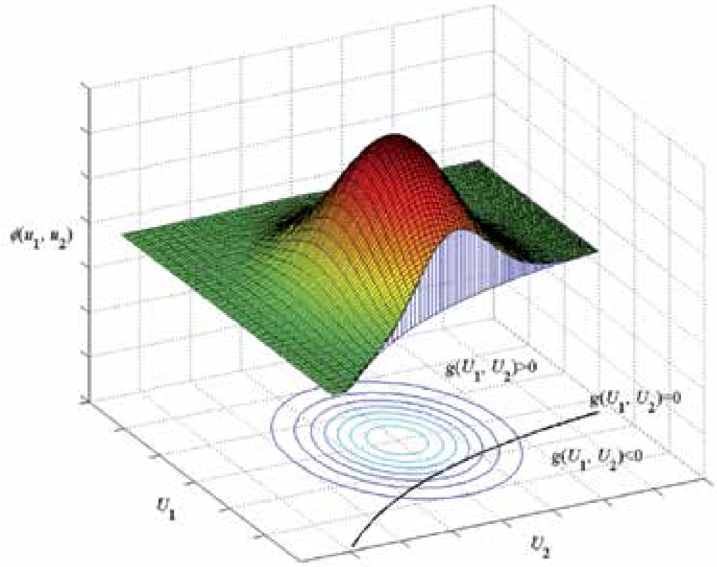


Fig. 3 - Space of uncorrelated standard normal variables: Rosenblatt transformation

Fatigue Analysis of Tubular Welded Joints

The aforementioned Ultimate Limit Strength criteria can be applied to the study of the fatigue failure of a component. In the present case the component is a welded tubular joint.

In a steel jacket type platform, tubular joints connect the main elements of the structure (bracing and legs) and represent, as always, the critical factor in relation to the fatigue behavior. As for the latter, it has a significant impact in all structures characterized by persistent and extreme dynamic loads (North Sea), or when there are no dominant high load design events such as hurricanes or typhoons (Arabian Gulf, West Africa).

The importance of fatigue as a key design consideration has been emphasised by the increasing use of higher strength steels and welded connections. In the earlier offshore platforms, steel strengths were low and the connection were riveted or bolted. These resulted in larger member cross-sections, highly redundant connections and lower cyclic stress. Since the fatigue strength of steel is not strongly correlated with its yield strength, fatigue was a lesser problem for the members and connections in such structures. This is not the case with more recent platforms that use steel yield strengths as high as 700 MPa Such designs typically deploy smaller member cross-sections and are inherently more vulnerable to fatigue failure.

Limit State Criteria are based on experimental data characterising SN-curves, and on the Miner-Palmgren hypothesis (the linearity of cumulative damage):

$$g(t) = \Delta - D \quad and \quad \mathrm{D} = \sum_{\mathrm{i}} \frac{\mathrm{n}(s_\mathrm{i})}{\mathrm{N}(s_\mathrm{i})}$$

where Δ describes the damage at fatigue failure; D is the cumulated damage up to the actual time t and it is defined by the ratios between the stress cycles number n with amplitude si applied to the member, and the stress cycles number N with the same amplitude needed to lead it to failure.

Instead of the SN-curve approach (Miner's rule) a fracture mechanics approach (Paris' law) needs to be adopted to assess more accurately the different stages of crack growth, including the calculation of residual fatigue life beyond through-thickness cracking.

$$g(t) = R(a_c, a_0) - S(t, t_0) = \int_{a_0}^{a_c} \frac{da}{\left[Y(a)\sqrt{\pi a}\right]^m \cdot G(a)} - CA^m\Gamma\left(\frac{m}{B}+1\right)v_0(t-t_0)$$

The above expression defines the difference between the crack growth from ao to ac (resistance), and the crack growth corresponding to a number of cycles No to N (load effect). These terms depend on the following parameters:

- Y(a) is the crack geometry function;
- C and m are material parameters;
- v0 is the average stress cycle frequency, so that v0(t-t0)=N-N0;
- $E[S^m] = A^m\Gamma\left(\frac{m}{B}+1\right)$ is the expected value of the long-term stress range, from which the function G(a) depends and the major uncertainties are related to.

It is crucial that the fracture mechanics approach is calibrated to the SN-approach for the initial stage of the fatigue life, to ensure that the initial crack size and the local geometry are properly represented.

Generally, while the fracture mechanics approach is used for inspection planning, the S/N approach is used for the standard fatigue analysis of tubular joints, which involves the following steps:

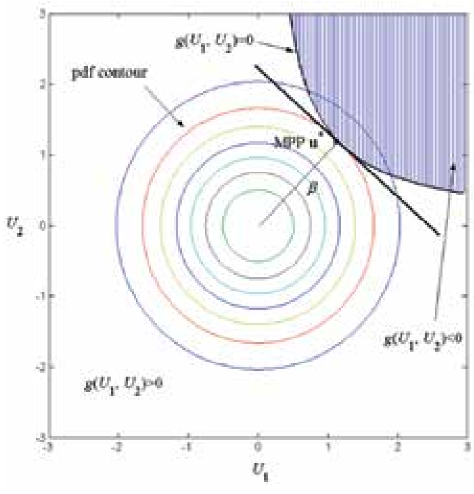


Fig. 4 - Graphical representation of the reliability index

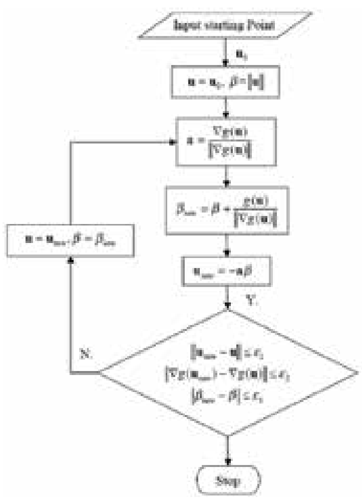


Fig. 5 - Hasofer-Lind and Rackwitz-Fiessler (HLFR) algorithm

H_m Significant Wave Height (ft)	T_n Zero Crossing Period (Seconds)								Total Probability
	0-2	2-4	4-6	6-8	8-10	10-12	12-14	14+	
0-3	0.114	0.225	32.551	16.810	0.105	0.001	0.000	0.000	49.806
3-6	0.000	0.002	7.951	30.110	2.575	0.003	0.000	0.000	40.641
6-9	0.000	0.001	4.255	3.250	0.006	0.006	0.000	0.000	7.518
9-12	0.000	0.000	0.080	1.450	0.150	0.003	0.001	0.000	1.684
12-15	0.000	0.000	0.000	0.006	0.125	0.075	0.015	0.000	0.221
15-18	0.000	0.000	0.000	0.000	0.050	0.010	0.010	0.000	0.070
18-21	0.000	0.000	0.000	0.000	0.010	0.045	0.000	0.000	0.055
21-24	0.000	0.000	0.000	0.000	0.000	0.005	0.000	0.000	0.005
24-27	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
27-30	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
30-40	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
40+	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
Total Probability	0.114	0.228	44.837	51.626	3.021	0.148	0.026	0.000	100.000

Table 1 - Two-dimensional scatter diagram for significant wave heights and zero crossing periods: P(Hm,Tn)

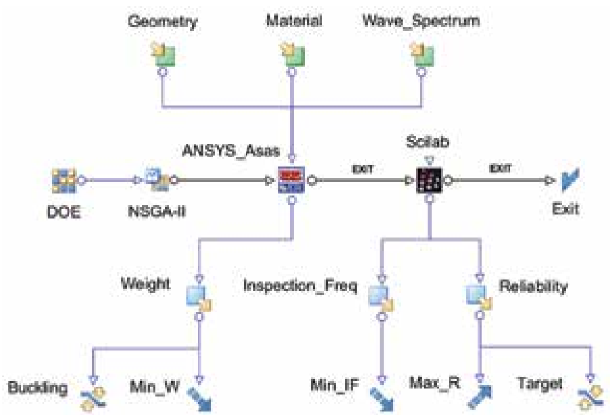


Fig. 6 - Sample modeFRONTIER workflow for multi-objective optimization

1. Prediction of the fatigue design wave parameters and their likelihood of occurrence (see Table 1)
2. Calculation of the nominal cyclic stress ranges and number of cycles at the brace ends
3. Calculation of the stress concentration factors (SCF) and the hot-spot stresses range (HSSR)
4. Selection of the applicable s-n fatigue design curve
5. Calculation of the fatigue damage ratio (miners fatigue damage ratio)
6. Demonstration

System Reliability Analysis and Failure Modes Identification

The main applications of structural reliability analysis can be summarized as follows:

- Optimization of design solutions, e.g. the fatigue life factor can be estimated and combined with the frequency of in-service inspections so that an expected lifetime cost is minimized. See Fig.6.
- Reliability updating of a structure based on new information obtained during the service life. In this context, reliability methods can also be an efficient tool for assessment of lifetime extensions of structures beyond their original design lives.
- Inspection planning at the design stage. Here, the probability of a fatigue crack is linked to the probability of detecting a crack in a considered structural detail. Reliability methods are used to estimate the time to first inspection and to determine the interval between subsequent inspections.

The structural system reliability analysis plays a crucial role in each of the aforementioned applications. For a structure with a high degree of redundancy, the failure mode approach is used to evaluate system reliability. That is, all the potential failure modes (mechanisms) in a structure are modeled as elements of a series system (MCS, Minimum Cut Sets representation). See Fig.7.

An offshore structure has numerous failure modes, but only some of them actually contribute to the system reliability, while others have very low probability of occurrence. Depending on the complexity of the structure and the accuracy required, several methods have been developed for the identification of the dominant modes, which can be divided into three basic categories:

1. Simulation-based approaches (directional simulation)
2. Plasticity-based approaches (β-unzipping method and linear programming)
3. Enumeration approaches (branch and bound method and incremental loading method)

The branch and bound is the most commonly used method in the offshore field. See Fig.8-9.

The branch and bound method uses a probabilistic research algorithm for the identification of failure paths, i.e. the failure paths are enumerated depending on their probability of occurrence. In the process of path enumeration, many paths have common elements and are highly correlated.

For the above reasons, the branch and bound method, although theoretically rigorous, is very expensive for large structures. To speed up the search, various deterministic approaches are used (incremental techniques and plastic mechanisms analysis), by replacing the random variables of stress and resistance by their characteristic values.

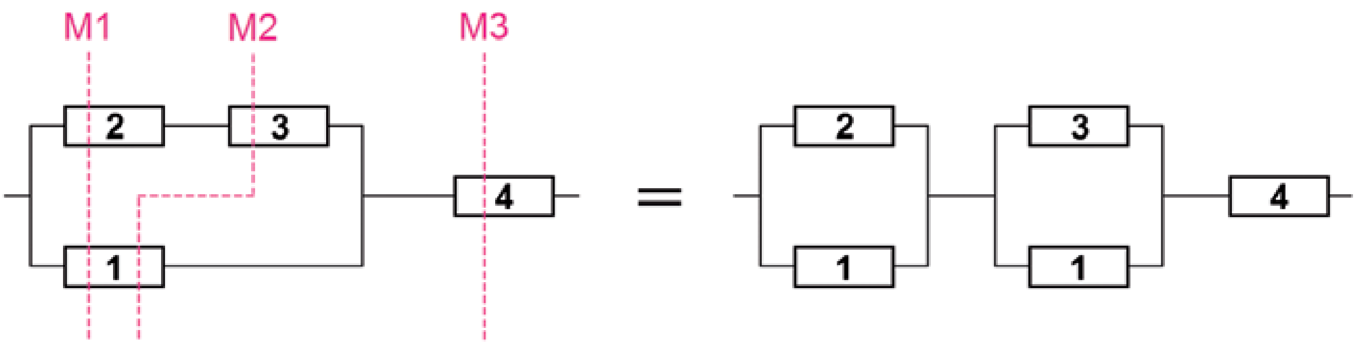


Fig. 7 - Minimum cut set representation of a system with three failure modes

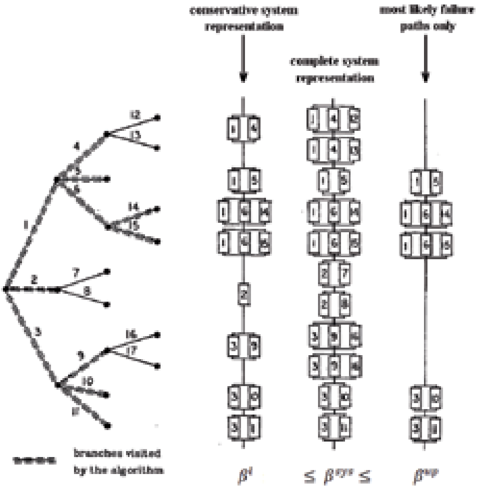


Fig. 8 - Truncated enumeration of dominant failure modes through branch and bound method.

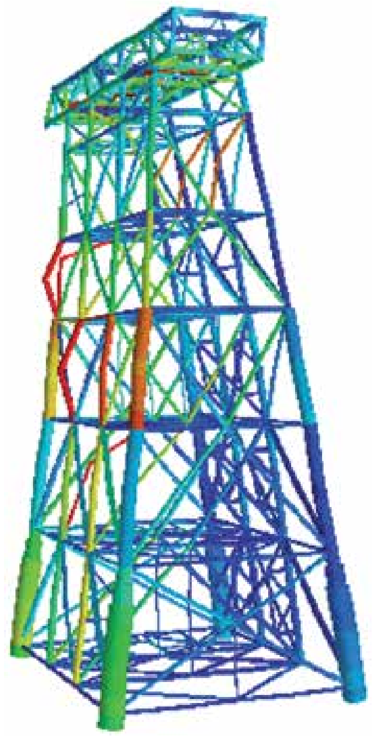


Fig. 9 Dominant failure modes in a steel jacket.

These methods are faster, but do not ensure the effective identification of all probabilistically dominant modes. To combine the demand of accuracy and computational efficiency, research undertaken in the last decade has addressed the use of genetic algorithms.

Reliability Updating and Inspection Planning

Although the main application of reliability based methods has been in requalification of existing offshore structures, it is also feasible to apply fatigue reliability analysis at the design stage to optimize inspection, maintenance and repair (IMR) strategies by using an event tree procedure (event 1 = "crack detection and repair", event 0 = "no crack detection", see Fig.10).

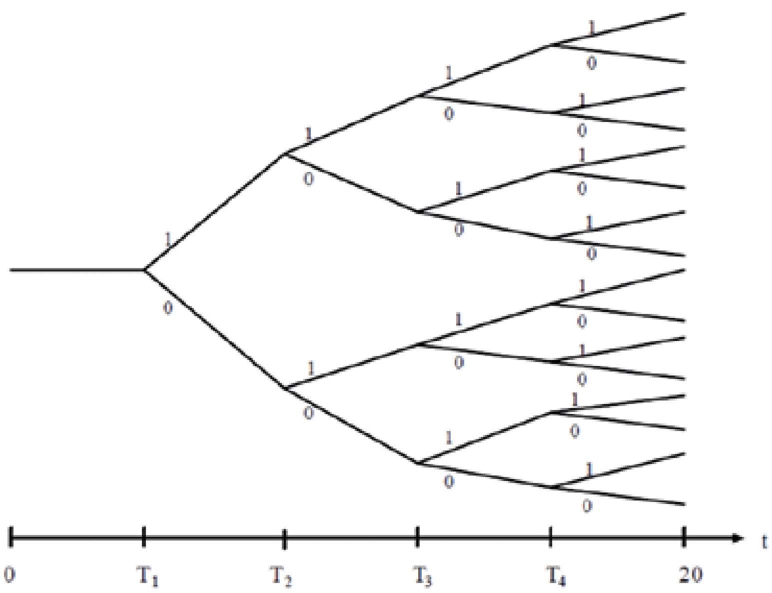


Fig. 10 - Fig. 10 . Inspection and repair strategy for event tree analysis considering 4 consecutive inspections with a frequency of 4 years

Given the inspection event IE, the fatigue failure probability of a joint can be updated based on the definition of the conditional probability,

$$P[g(t) \le 0 \mid IE] = P[(g(t) \le 0) \cap IE] \,/\, P(IE)$$

where g(t) is the fatigue limit function according to Paris' law.

The inspection event IE is defined by an expression similar to that of g(t), where the upper limit of integration ac in the resistance term is replaced by the minimum detectable crack size aD, and the lower limit depends on the repairs history, as shown in Table 2.

This definition implies that IE is positive when the crack size developed up to the time of inspection is smaller than aD, thus, the event IND = IE>0 means "no crack detection" while the complementary event ID = IE<0 stands for "crack detection and repair".

$$IE = R(a_2, a_1) - S(T_b, T_a) \begin{cases} > 0 & \textit{no crack detection} \\ < 0 & \textit{crack detection and repair} \end{cases}$$

The fatigue failure probability of a joint can be evaluated using the law of total probability; e.g. referring to Fig. 10, the probability of failure for time T1<t<T2 is given as follows:

$$\begin{aligned} P_f(t) = P_f(0, T_1) &+ P[g(0, T_1) > 0 \cap I_{ND} \cap g^0(T_1, t) < 0] \\ &+ P[g(0, T_1) > 0 \cap I_D \cap g^1(T_1, t) < 0] \end{aligned}$$

To perform an inspection implies the reduction of the uncertainty associated with the knowledge of the system. In mathematical terms, this fact is reflected in Fig.11 by an increase of the index of reliability β both in the case of in-service inspection (green curves), and in the case of inspection planning at the design stage (red curve). Although the latter case has a smaller effect on the damage curve, it plays an important role in reducing the construction costs. In Fig.12 is shown the effect of inspection planning at the design stage on the reliability index trend: the design solution given by red curve satisfies the constraint on the allowable cumulative damage (Δd = 0.1), despite the member is the result of a non-conservative design (it is designed to reach a cumulative damage Δd = 0.2 if no inspections are carried out). This results in a lower cost design.

Reliability updating and inspection planning cannot be divorced from system reliability analysis, as the failure of redundant offshore structures may result in a sequence of overload or fatigue failures of components. Since the

Time	Branch	Fatigue Limit Function		Inspection Event	
		Damage func. $R(a_2,a_1)$ *	Time limits T_b, T_a **	Damage func. $R(a_2,a_1)$ *	Time limits T_b, T_a **
0< t <T1		a_c, a_0	t, T_0	a_{D1}, a_0	T_1, T_0
T1< t <T2	1	a_c, a_{R1}	t, T_1	a_{D2}, a_{R1}	T_2, T_1
	0	a_c, a_0		a_{D2}, a_0	T_2, T_0
T2< t <T3	11	a_c, a_{R2}	t, T_2	a_{D3}, a_{R2}	T_3, T_2
	10	a_c, a_{R1}		a_{D3}, a_{R1}	T_3, T_1
	01	a_c, a_{R2}		a_{D3}, a_{R2}	T_3, T_2
	00	a_c, a_0		a_{D3}, a_0	T_3, T_0
T3< t <T4	111	a_c, a_{R3}	t, T_3	a_{D4}, a_{R3}	T_4, T_3
	110	a_c, a_{R2}		a_{D4}, a_{R2}	T_4, T_2
	101	a_c, a_{R3}		a_{D4}, a_{R3}	T_4, T_3
	100	a_c, a_{R1}		a_{D4}, a_{R1}	T_4, T_1
	011	a_c, a_{R3}		a_{D4}, a_{R3}	T_4, T_3
	010	a_c, a_{R2}		a_{D4}, a_{R2}	T_4, T_2
	001	a_c, a_{R3}		a_{D4}, a_{R3}	T_4, T_3
	000	a_c, a_0		a_{D4}, a_0	T_4, T_0
T4 <t	1111	a_c, a_{R4}	t, T4		
	1010	a_c, a_{R3}			
(examples)	0101	a_c, a_{R4}			
	0000	a_c, a_0			

$$^{*}\,R(a_2,a_1)=\int_{a_1}^{a_2}\frac{da}{Y(a)^m\,(\pi a)^{m/2}\,G(a)} \qquad {}^{**}\,S(T_b,T_a)=C\upsilon_0\left(T_b-T_a\right)A^m\Gamma\left(1+\frac{m}{B}\right)$$

Table 2 Integral limits in fatigue limit function and IE function for the event tree defined in Fig.10

updating of systems reliability by inspection events may imply lengthy calculations, simplifications should be implemented, e.g. by updating the failure probability of non-inspected joints based on results of inspected joints.

This simplification is justified due to correlation between joints in complex systems, which implies similarities in load effects and resistance among the components. Thus updating the reliability of non-inspected joints may be carried out by utilizing inspection results from inspected joints and so the overall system reliability may be updated accordingly.

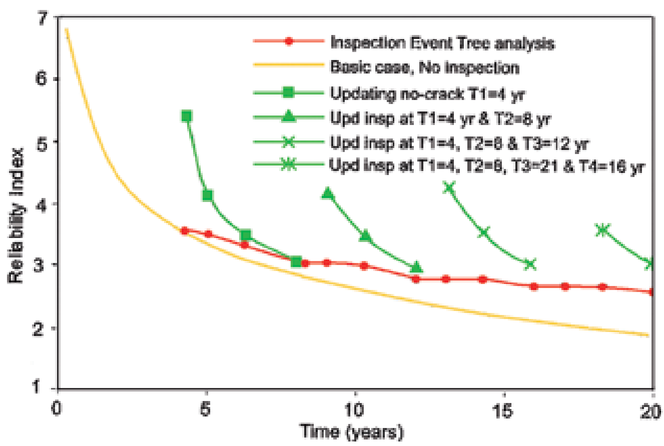


Fig. 11 - Reliability updating of a tubular joint considering 1, 2, 3 and 4 inspections with no crack found carried out at 4, 8, 12 and 16 years, respectively, compared with updating through event tree analysis

Conclusions and Future Research

The concept of component reliability was introduced as the cornerstone of the reliability assessment of the entire system. In particular, its importance was emphasized with reference to the issue of inspection planning. Work is underway to implement operations research strategies to identify the dominant failure modes in the structure (e.g. branch and bound) and incorporate, within these methodologies, numerical procedures for the efficient probabilistic assessment of reliability (eg. FORM).

This code will be fully developed in Scilab and integrated into a routine to calculate the reliability of offshore structures, based on their stress responses computed by the ANSYS ASAS software, linked to the optimisation of inspection planning through the modeFRONTIER software tool.

M.N. Coccon, U. Galvanetto
Dipartimento di Ingegneria Industriale,
Università di Padova - Italy

Livio Furlan, EnginSoft

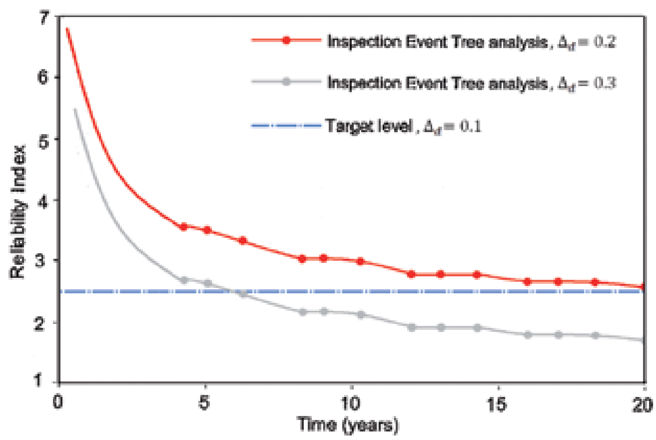


Fig. 12 - The allowable cumulative damage Δd in design can be relaxed when inspections are carried out: the target level Δd=0.1 with no inspections can be reached by a Δd=0.2 when inspections are carried out every fourth year

Combined 1D & 3D CFD Approach for GT Ventilation System analysis

The Gas Turbine ventilation system is designed to supply the necessary amount of air for cooling and to prevent the accumulation of hazardous gases in the enclosure by maintaining a slight over-pressure. The classical GE approach to studying ventilation system operating conditions consists of modeling the whole system as a series of discrete losses, where the ASHRAE duct-fitting database provides the corresponding pressure loss coefficients. The system is solved by means of a one-dimensional flow simulation tool (Flowmaster).
The goal of this work was to improve the critical points that affect the above-mentioned procedure, such as modeling of complex fittings and bend interactions. For this purpose, dedicated CFD analyses were performed to characterize the loss coefficient for splitting and bend interactions at different operating conditions (split percentage and inlet flow rate) for two different ventilation systems. The resulting loss coefficient curves have been implemented within the corresponding one-dimensional Flowmaster models. Finally, to characterize off-design conditions, a variable Heat Rejection model (obtained from previous CFD analyses) and real fan curves were used.

This new approach produces more accurate results, as confirmed by the close agreement with experimental measurements. Among the benefits of using this new approach is the ability to characterize the flow behavior of complex fittings. This would be useful in the event of a fitting redesign or for noise reduction analyses.

Current GE approach to studying Gas Turbine Ventilation Systems

A ventilation system must provide a continuous source of cooling air over the entire Gas Turbine operation range in order to:

- maintain a uniform and constant airflow through the flange-to-flange Gas Turbine at all ambient conditions;
- remove heat and maintain the air temperature in the compartment below the operating limit. (The operating limit is set according to the temperature rating of the components located in the compartment);
- eliminate stagnation zones and prevent the accumulation of hazardous gases;
- prevent the ingress of dust and sand in gas turbines located in regions prone to sandstorm conditions by means of proper compartment pressurization.

Specific Design Practices provide a general description, acceptance limits and design criteria that a ventilation system must meet for Oil&Gas applications (e.g., enclosure design temperature ranges, design pressure ranges, purging ranges, etc.).
As mentioned, the current GE approach to studying GT ventilation systems consists of modeling the whole system as a series of “blocks”. Each block represents a source of pressure loss (concentrated loss) due to changes in shape (e.g., elbow, transition, etc.), flow direction or the presence of physical obstacles within the system. The ASHRAE duct-fitting database provides the corresponding pressure loss coefficients.
Following the net balancing by means of a one-dimensional flow tool (Flowmaster), the system is characterized in terms of velocities, pressures, and flow rate split.
Critical points for this approach are the modeling of complex fittings and bend interactions. In order to improve the current Ventilation System calculation procedure, dedicated CFD analyses were performed for these critical points. A combined 1D & 3D CFD approach was adopted to study two different GE Ventilation Systems, called for simplicity System A and System B.

Numerical calculations for System A

The current System A Flowmaster network, modeled as a series of discrete losses, is shown in Figure 1. The enclosure is modeled

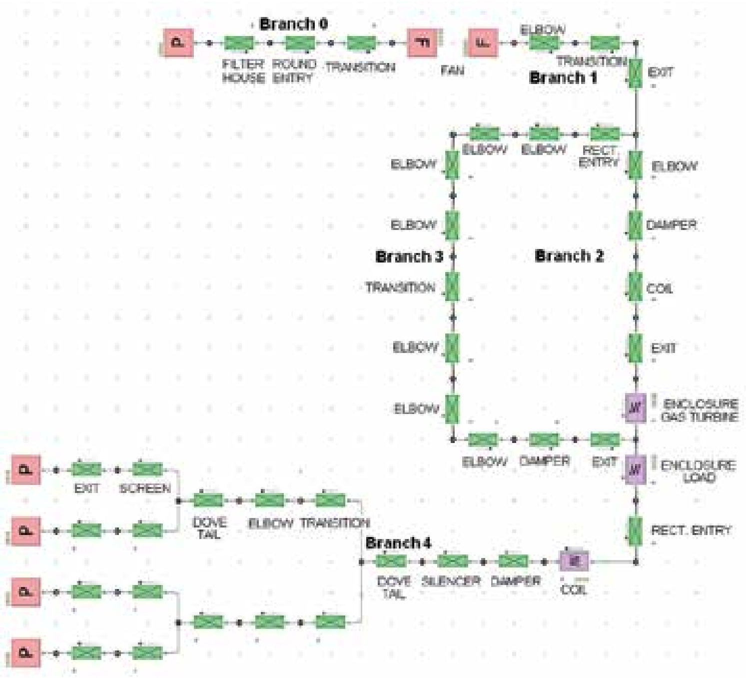


Fig. 1 - System A Flowmaster model based on discrete losses

as two heaters and the fan as two flow sources with a flow rate of 65000 m³/h, estimated by using the enthalpy balance equation:

$$\dot{m} = \frac{HR}{Cp(T_{out} - T_{in})}$$

were:

$\dot{m}$ = mass air flow [Kg/s],
HR = enclosure heat rejection [W]
Cp = specifiwec heat at constant pressure [J/Kg °C]
T_{out} = maximum allowable outlet air temperature [°C]
T_{in} = max ambient temperature [°C]

In order to develop a more suitable model (taking into account interactions, 3D characteristics of the fluid, etc.), dedicated ANSYS FLUENT CFD analyses were performed. In particular, a critical point for the discrete losses modeling is the flow split into the Load Compartment and the Gas Turbine Compartment (see Figure 2).

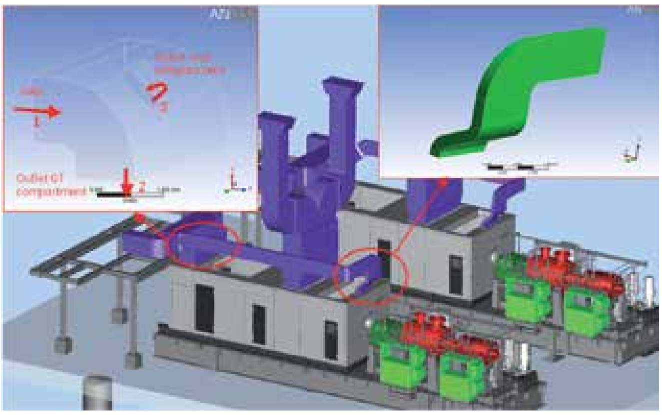


Fig. 2 - Analyzed split (left) and Load Compartment final section (right), System A

It is useful to define the coefficients K12 and K13 as:

$$K_{12} = \frac{P_{01} - P_{02}}{1/2\rho V_2^{\,2}} \; ; \qquad K_{13} = \frac{P_{01} - P_{03}}{1/2\rho V_3^{\,2}}$$

where:

P_{01} = inlet total pressure
P_{02} = GT Compartment total pressure
P_{03} = Load Compartment total pressure
V_2 = GT Compartment mean velocity
V_3 = Load Compartment mean velocity

For the characterization of the flow split at different operating points, two test campaigns were performed. In both cases the inlet flow rate was fixed (65000 m³/h and 130000 m³/h, respectively) and, for each of these, a variable split percentage between the GT and Load compartments was used.

	K12 (GT Compartment)	K13 Load Compartment
ASHRAE Database	0.4	0.71
modified ASHRAE model (experience based)	1.15	1.5
CFD	0.4-0.6	2.28

Table 1: Loss coefficients used for standard calculations, System A

These analyses provided K^{12} and K^{13}, defined in (2), as a function of the flow rate split (see Figure 3).
Subsequently, these coefficients were implemented within the corresponding one-dimensional Flowmaster model.

A comparison between the loss coefficients obtained using CFD and those coefficients used for standard calculations is summarized in Table 1.

Finally, to better simulate the ventilation system a bend interaction analysis was performed on the Load Compartment final section, which is highlighted in Figure 2 (for System B the geometry of this section is the same). The total loss coefficient as a function of the inlet velocity is shown in Figure 4. The loss coefficient decreases as the inlet flow velocity increases, and a good agreement with the ASHRAE database value was found for a velocity of about 5m/s. For higher velocity values the difference between the two curves (CFD and ASHRAE) starts to be significant. Again, the loss coefficient curve obtained was implemented within the new model.

The fan, previously modeled as two flow sources, was replaced by the "FAN" element with the corresponding real operating curve.
The final System A Flowmaster model including the main differences from the standard approach is shown in Figure 5.

The results obtained with the new model were compared with the results obtained by the ADV (Air Ducts and Ventilation) department using a model based on the ASHRAE loss coefficient with appropriate corrections based on experience and with the results obtained with a pure ASHRAE model (see Table 2). The reliability of each approach was evaluated through comparison with experimental data.

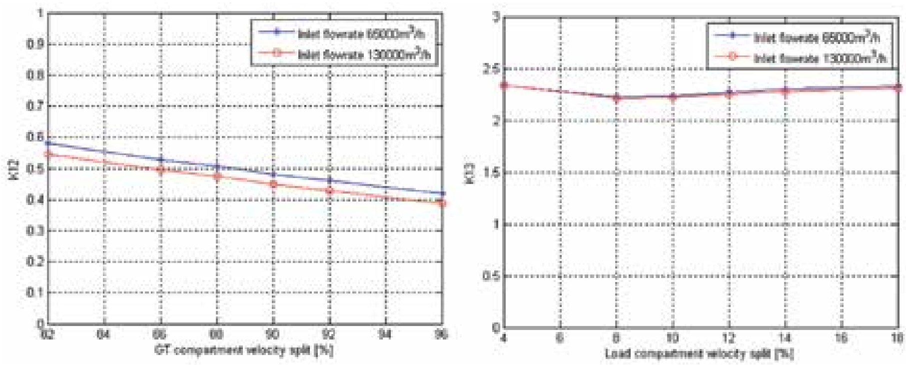


Fig. 3 - K_{12} and K_{13} as a function of flow rate split for two different inlet flow rates, System A

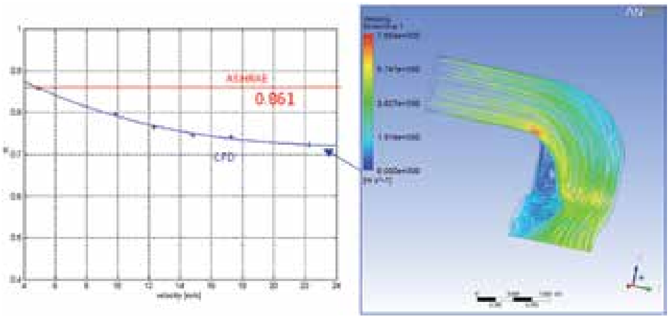


Fig. 4 - Loss coefficient curve for Bend Interaction

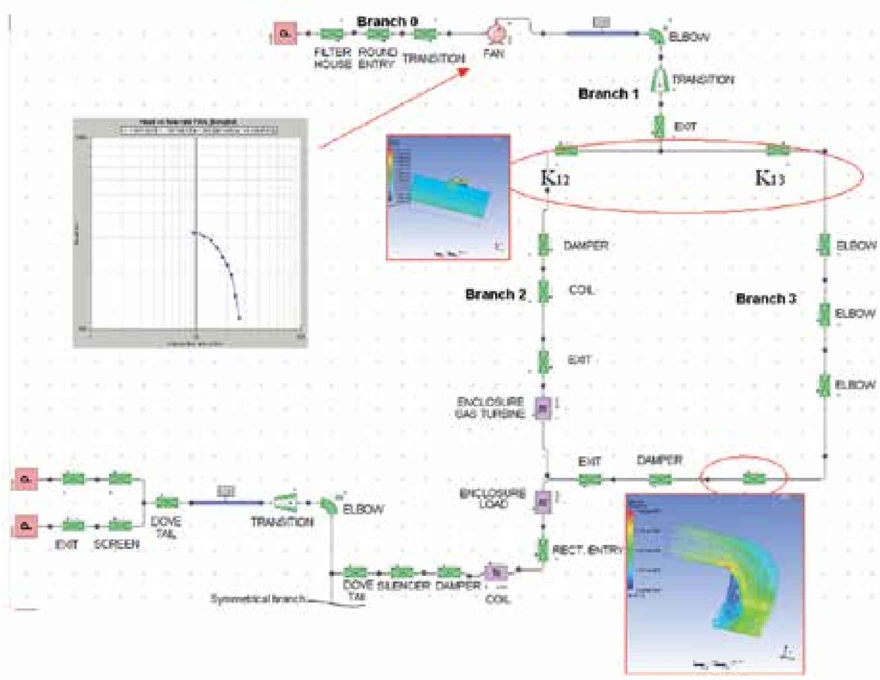


Fig. 5 - New System A Flowmaster model.

Numerical calculations for System B

Also for the System B split, several tests were performed to determine the split loss coefficients for different operating conditions. Figure 7 shows K^{12} and K^{13} as a function of the split flow rate percentage between the GT and Load compartments for an inlet flow rate equal to 70000 m^3/h (design flow rate). As one can see, both curves follow a linear trend.

Similar to the System A model, the new System B Flowmaster model contains the loss coefficient curves obtained from CFD analyses (including the bend interaction curve) and the real fan operating curve. Finally, in order to better simulate the heat removal, the heat rejection was modeled as a function of the mass flow rate, in accordance with recent studies performed by the SYS-OPT (System Optimization) department, that is:

$$HR = HR_0 \left(\frac{\dot{m}}{\dot{m}_0} \right)^n$$

where:
HR = heat rejection
HR^0 = reference heat rejection
$\dot{m}$ = mass flow rate
$\dot{m}_0$ = reference mass flow rate
n = reference exponent

The final System B Flowmaster model is shown in Figure 8. Figure 9 shows the GT and Load Compartment velocity obtained with the STD model (previous calculations) and the new model for dirty and clean filter house conditions. In both cases, the load compartment velocity obtained with the new approach is significantly higher than the old value

Table 2 summarizes the results obtained for the enclosure pressure. Using the new approach we got a favorable level of approximation with respect to the measured value (error equal to 7%). The other two approaches yielded errors higher than 25%.

Figure 6 shows for each model the load compartment velocity and the corresponding error from the measured value at clean filter house conditions. The measured mean velocity is 12.47 m/s.
Both the new model and the modified ASHRAE model (experience-based) led to a high level of agreement (error lower than 5%). On the contrary, the pure ASHRAE model produced an error of 18%.

	Enclosure Pressure[mmH2O]	Measured value[mmH2O]	Error[%]
Discrete loss model (ASHRAE)	54.40	43.0	26.5
Discrete loss model (Experience based)	54.86	43.0	27.6
New model (Flowmaster+CFD)	40.00	43.0	-7.0

Table 2: Enclosure pressure, clean filter house conditions, System A

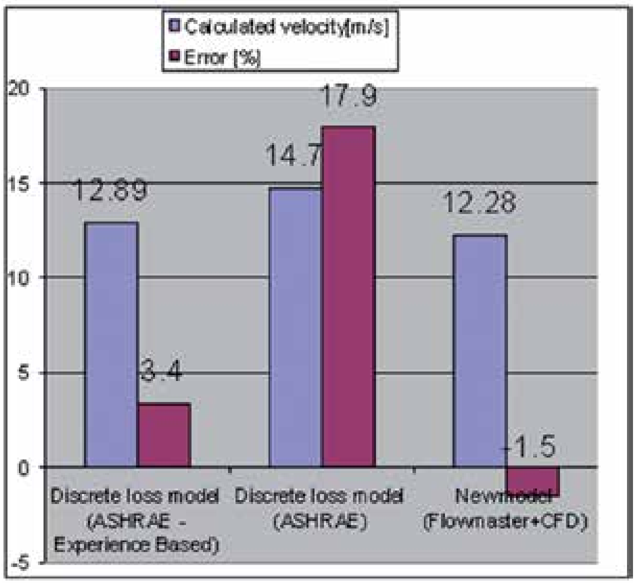


Fig. 6 - Load compartment velocity, clean filter house conditions, System A

(+49%). In particular, for the new approach, we got a split of 89-11% compared to a value of 92.7-7.3% obtained from previous calculations. Considering that the target flow rate is 90-10%, the new approach again provides more accurate results.

No significant variations between the two approaches in terms of enclosure pressure and temperature were found.

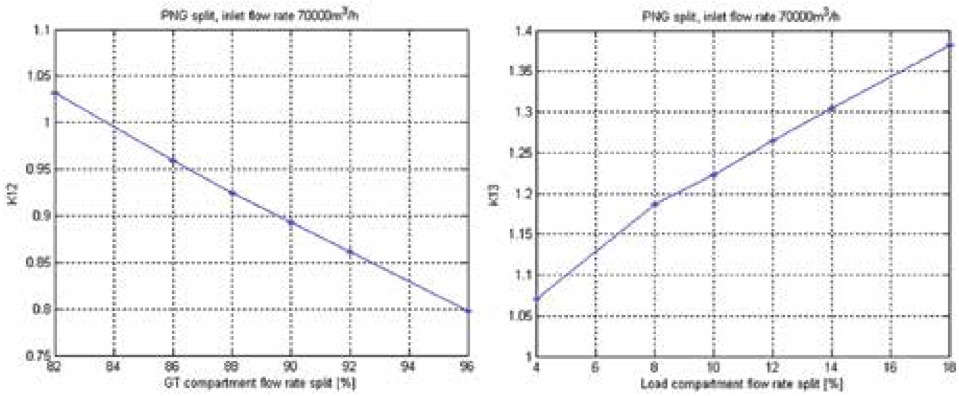


Fig. 7 - K_{12} and K_{13} as a function of flow rate split, System B

Conclusions

In this work, a combined 1D and 3D numerical approach was adopted to study two GE ventilation systems. This approach, compared to the current one-dimensional approach, improves the simulation of the actual operating conditions in terms of inlet flow rate, duct velocity and enclosure pressure, as confirmed by the close agreement with experimental measurements.

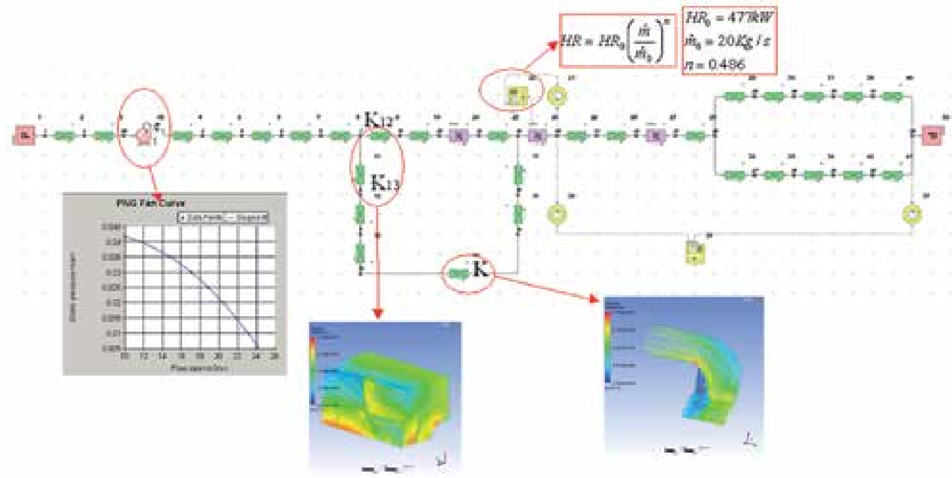


Fig. 8 - New System B Flowmaster model

Among the benefits of using this new approach is the ability to characterize the flow behavior of complex fittings. This would be useful to support the redesign of fittings or for noise reduction analyses.

L. Barbato, M. Blarasin, S. Rossin
GE Oil&Gas

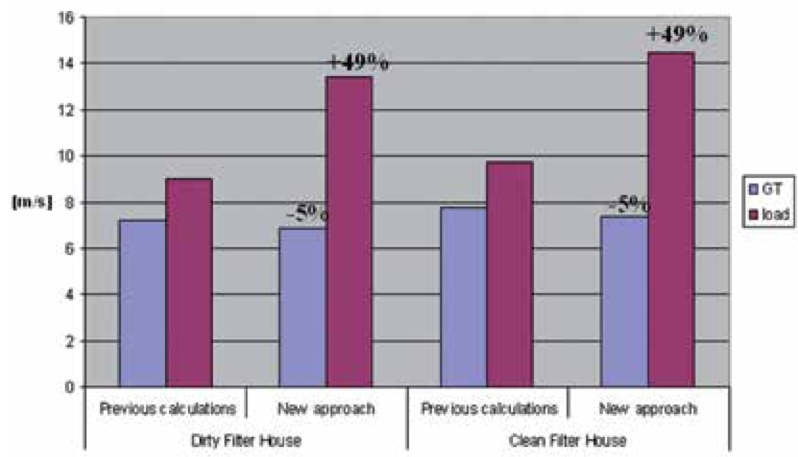


Figure 9 - GT and Load compartment velocity for STD and New approach, System B

About GE Oil&Gas

GE Oil&Gas (www.ge.com/oilandgas) is a world leader in advanced technology equipment and services for all segments of the oil and gas industry, from drilling and production, LNG, pipelines and storage, to industrial power generation, refining and petrochemicals. We also provide pipeline integrity solutions, including inspection and data management, and design and manufacture wire-line and drilling measurement solutions for the oilfield services segment. As part of our 'Innovation Now' customer focus and commitment, GE Oil&Gas exploits technological innovation from other GE businesses, such as Aviation and Healthcare, to continuously improve oil and gas industry performance and productivity. GE Oil&Gas employs more than 12,000 people worldwide and operates in over 100 countries.

A CFD Methodology for Modelling the Cool Down Process in a 3-Phase Subsea Separator Tank

Over 35% of the world's total production of petroleum comes from subsea exploration with deepwater production the only sector that has continued to expand after 2012 (Neal Prescott, 2012). This makes research into reliable and more efficient subsea equipment both desirable and essential. This necessity is particular strong in Brazil where 90% of oil production occurs off-shore (US Energy Information Administration - EIA, 2013).

FMC Technologies in Brazil has a large research center focused on increasing the recovery factor for oil production. A common maxim is that if part of the required separation is performed subsea, then topside processing capacity can be reduced resulting in more effective production. This can contribute to both increased revenues and a better environment.

To optimize the design and maintenance process of subsea separators FMC Technologies worked together with ESSS, an ANSYS channel partner in Brazil, who developed the best approach to accurately simulate a pause period in the equipment operation, the cool down process. High consideration was given to the accuracy of the results required, as well as for the run time and computational resources available.

Separators are commonly subjected to convection from cold sea currents and during the cool down process the previously heated internal fluids (e.g. water, oil and natural gas) can reach temperatures low enough to cause the formation of hydrates. This is a major concern as hydrates can cause blockages in the production line, hampering production and decreasing operator profits.

The specific case studied by ESSS consisted of a 3-phase gravitational separator with a structural steel frame and an insulation layer and having a 6 hours cool down period. No entrances and exits were considered in the tank model, therefore all movements were due to natural convection given the fluids were initially completely segregated and stagnant. The external temperature was 4°C with the separator set at 50°C at the beginning of the process. ANSYS CFX was employed to test varying simulation approaches including different mesh refinement levels (2.3 million elements and 500 thousand elements) and physics 'simplifications'. The following results were compared; general physics behavior, average and minimum temperature and simulation time.

In addition to a 3-phase model comparison the physical 'simplifications' included a 2-phase model. This consisted of approximating both liquid phases as one single phase, with thermal properties defined as the average of water and oil weighted by the volume fraction. A 1-phase 'simplification' involved the simulation of only one phase, with all thermal properties averaged by the volume fractions. A third 'simplification' involved a conduction-only model, developed by considering that any movement caused by natural convection could be neglected, and therefore each phase was represented as a 'solid material'.

The 3-phase model comparison between coarse and fine meshes showed that the coarse mesh was a good cost-effective model size delivering conservative results when compared to the fine mesh simulation with regards to the possibility of hydrate formation. The coarse mesh model

	Average Temperature	Difference [%]	Minimum Temperature	Difference [%]	Simulation Time
Base 3-Phase Model	41.9°C	-	22.2°C	-	16 days
2-Phase Model	42.2°C	0.7	22.8°C	2.7	13 days
1-Phase Model	42.6°C	1.7	23.1°C	4.1	5 days
Heat Conduction Model	46.8°C	11.7	11.3°C	49.1	30 hours

Fig. 1 - Results for each 'simplification' in comparison to the base 3-phase case

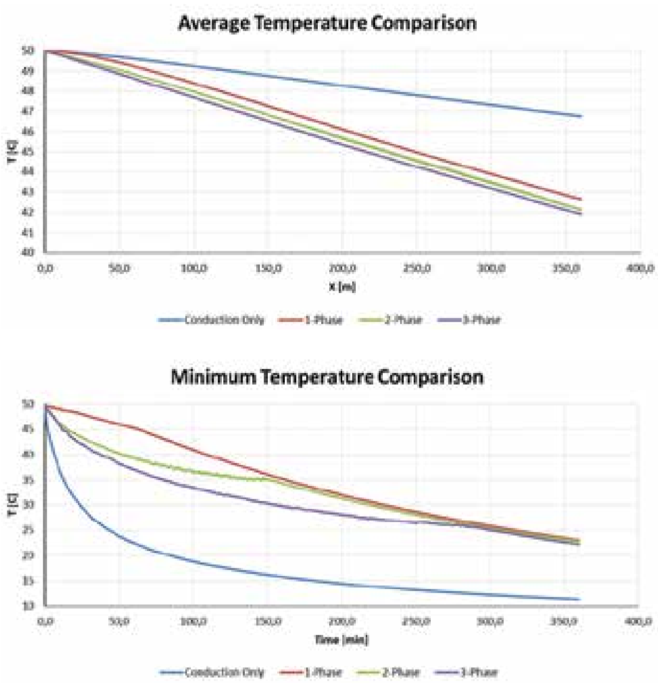


Fig. 2 - Average (a) and minimum (b) temperature over time

always showed smaller temperature values, the minimum temperature for being 10% lower than the refined mesh (around 3°C of difference). Operational data belonging to FMC Technologies showed excellent agreement between the 3-phase approach and the separator behavior during cool down. Table 1 and Figure 1 show the results for the physical 'simplifications' using the coarse mesh. The 3-phase approach is recommended for more accurate results. The 1-phase case delivered similar results for average and minimum temperature compared to 3 and 2-phase cases, but the temperature profile was very different. This approach should therefore be used only to achieve initial estimates or faster results.

*Fernando F. M. C. de Guamá, Patrick Farias - **FMC Technologies***
João Américo Aguirre Oliveira Jr., Amanda Briggs da Silva,
Clarissa Bergman Fonte, Thiago Felippe Ribeiro
ESSS – Engineering Simulation and Scientific Software
*Mike Chudiak - **ANSYS Inc***

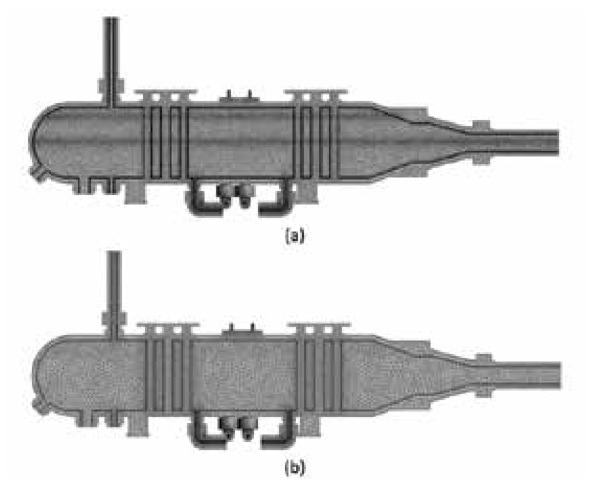


Fig. 3 - Refined mesh (a) and coarse mesh (b)

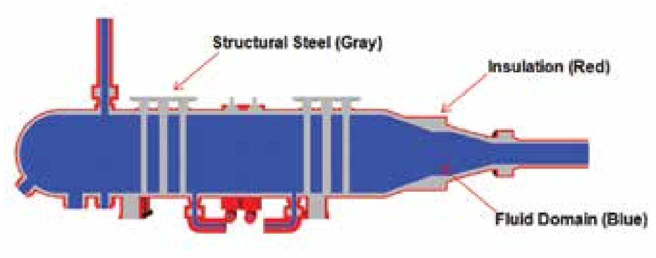


Fig. 4 - Separator computational model

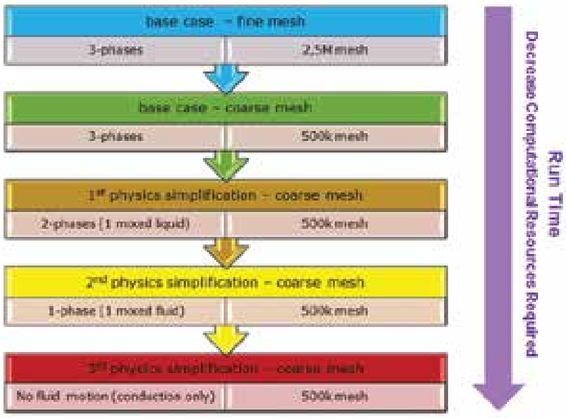


Fig. 5 - Modeling approaches tested

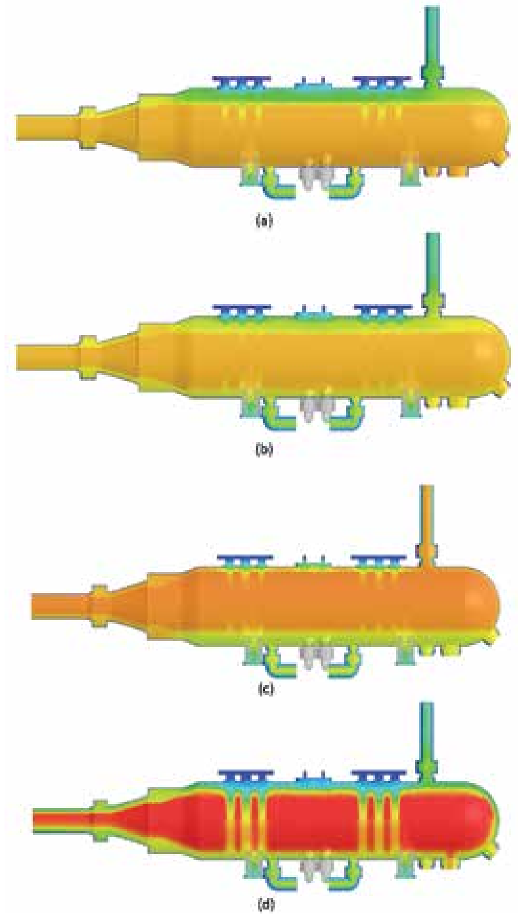


Fig. 6 - Temperature profile for the base 3-phase case (a); the 2-phase simplification (b); the 1-phase simplification (c); and the heat conduction model (d)

A simulation tool for the Dissociation of Hydrate Plugs in horizontal lines

Due to the combination of low temperatures and high pressures that occur in the seabed the presence of natural gas and water inside pipelines can lead to solid hydrate formation. As seen in Figure 1 this may impair or even block the flow of produced fluids. One of the procedures used to dissociate the hydrate plug is line depressurization, in one or both sides of the line. This forces operational conditions (temperature and pressure) outwith the hydrate equilibrium envelope, therefore dissociating the hydrate plug.

The problem is an important flow assurance issue and one of major concern at Petrobras. For this reason, ESSS, the ANSYS channel partner in Brazil, worked in tandem with the Petrobras Research Center to develop a simulation tool for modeling the dissociation of a hydrate plug in a horizontal line considering the line depressurization process. In developing this model both theoretical and experimental data were used.

The key parameter to manage the plug remediation in one-sided depressurization is the dissociation time, defined here as the time needed to observe a drop in the upstream pressure related to the mass flow through the annular channel that forms between the pipe wall and the hydrate plug.

The simulation tool was developed in ANSYS Fluent, version 15.0.7 and using the ANSYS Workbench platform to implement a parametric model. This allows the tool to analyze the phenomenon for different geometries and/or operating conditions by easily changing the project parameters.

The modeling strategy considered the hydrate plug as a homogeneous and isotropic porous medium. Heat transfer was modeled for convection in the environment and conduction through the pipe wall and insulation, which adds the necessary energy for the plug dissociation. Figure 2 shows the simulation domain that was considered in this study.

The work demanded the development of user defined routines in ANSYS Fluent. At the end of each time step the number of cells at a temperature above the equilibrium temperature for a required pressure was calculated, and latent heat consumed during the dissociation of the hydrate in each cell was accounted as a sink in the energy equation. Thereafter, the mass of dissociated

Fig. 1 - Hydrate plug

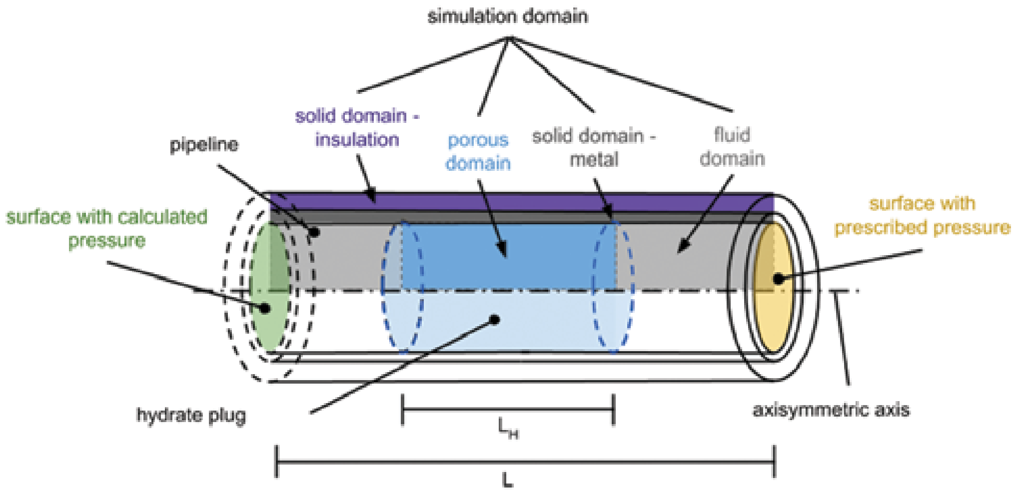


Fig. 2 - Computational domain used in the simulation

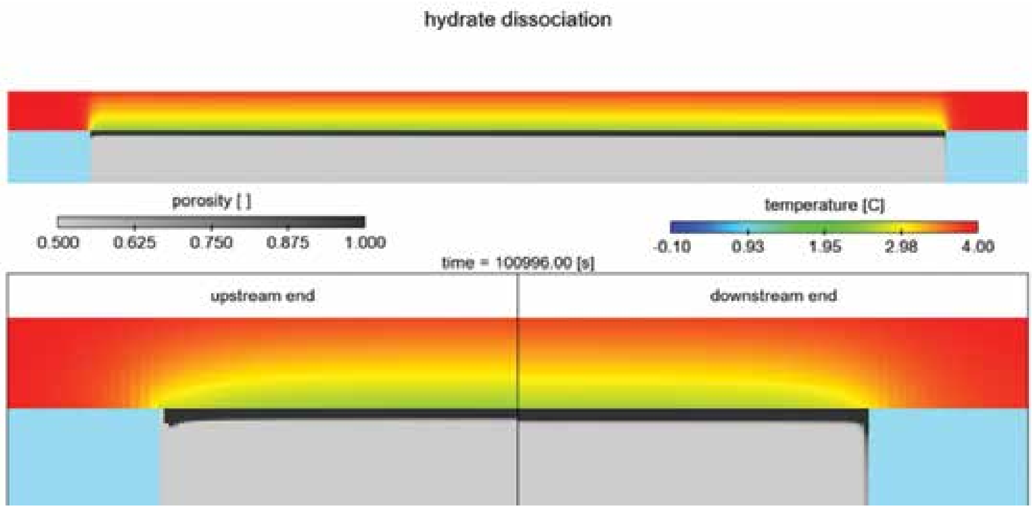


Fig. 3 - Porosity and temperature profiles for unilateral depressurization of the line

hydrate was calculated and the local variation in the porosity and permeability of the cell obtained.

Using this methodology, it was possible to replicate the process of plug dissociation in one-sided depressurization operations. Figure 3 shows the porosity profile (gray scale) and temperature profile (rainbow scale) for the plug, as well as for its downstream and upstream regions. It shows that dissociation occurs preferentially in peripheral regions of the plug, since such cells receive additional heat for dissociation due to the heat conduction and convection through the wall to the external environment.

The model was able to adequately represent the process of hydrate plug dissociation until the formation of an annular channel and it can be used to assess this dissociation time for different operation conditions and/or geometries.

João Américo Aguirre Oliveira Jr., Lucilla Coelho de Almeida,
Clarissa Bergman Fonte, Carlos Eduardo Fontes
ESSS – Engineering Simulation and Scientific Software
Flávio Barboza Campos, Leandro Saraiva Valim, Adriana Teixeira
CENPES/Petrobras - Petrobras Research Center

Petróleo Brasileiro S.A - Petrobras is a Brazilian company, headquartered in Rio de Janeiro, leader in the exploration, production, refining, transportation and marketing of oil products. It's one of the major Brazilian companies and one of the 15 most important oil industries in the world. It has a cutting-edge technology able to produce in deep and ultra-deep waters, with a world record of 2 Km. The oil daily output in 2013 has overcome the 2 million barrels.
http://www.petrobras.com

CFD Evaluation of a New Burner Arrangement inside a Refinery Single-Fired Furnace

The way that most industrial processes obtain energy is through combustion systems such as furnaces or boilers. Much effort has been made to understand the behaviour of these types of systems that involves different types of phenomena which are directly or indirectly connected. In this context combustion reaction is the major phenomenon where both kinetics and the mixing degree, in other words, turbulence effects, influence the flame behaviour. The resulting reaction produces a large amount of heat, leading for example, to buoyancy effects caused by great differences in temperature along the equipment. This makes it necessary to model all phenomena involved in the process from the relationship between turbulence and chemistry to density differences and buoyancy effects which are important to represent flow behaviour.
Within Petrobras's refineries crude oil is transformed in a several products such as gasoline, diesel, lubricants, aviation kerosene, naphtha, et al. Petrobras operates 12 refineries in Brazil and processes 2.1 million barrels of oil per day (bpd) with the potential to expand this amount to 3.3 million bpd by 2020. To do that it will be necessary to either create new refineries or extend the existing ones. This takes a lot of effort and investment and all new projects are designed at the Petrobras Research Centre (CENPES/ Petrobras).
Furnaces can be categorized amongst the main equipment within a refinery route process, particularly as they can perform several roles. One of these is heating the crude oil that enters the refinery in order to promote the distillation process. Typically, the burners are located towards the base and the tubes, where the crude oil flows, are located near the walls of radiant sections and inside the convective section.
To expand refinery capacity it is necessary increase the capacity of specific equipment. This clearly includes furnaces, whereby larger volumes allow bigger batch amounts of crude oil to be heated in order to experience the distillation process. CENPES/Petrobras worked with ESSS, the ANSYS channel partner in Brazil, on a CFD study to expand the capacity of a single fired furnace by including a new tube bundle between the radiation chamber and the convective zone. It was suggested that fire heaters were included on the ceiling of the radiation chamber due to the necessity of heat load increase by the inclusion of the new tubes, as is shown in Fig. 1.
In order to simplify problem solving, only the region with the new tubes was evaluated, given that the radiation chamber had already been simulated in previous studies, hence these previous results were used as boundary conditions on the domain inlet. To simulate the convection tubes without discretize them, the tubes region were simulated as a porous medium with pressure drop and heat loss equivalent to the convection tubes.
Two different types of burner configuration were considered; one with 10 holes for gas injection and another one with 16 holes (see

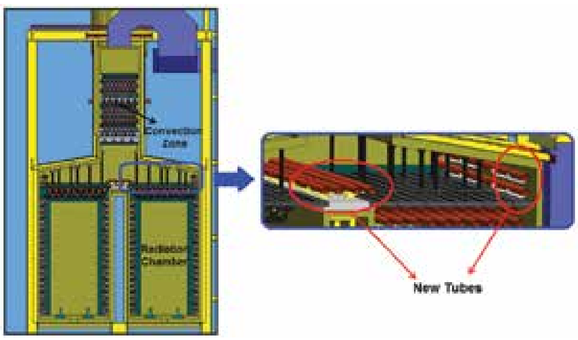


Fig. 1 - Furnace's design with the new tubes included

Fig. 2 - Fire heaters: a) 10 holes for gas injection; b) 16 holes for gas injection

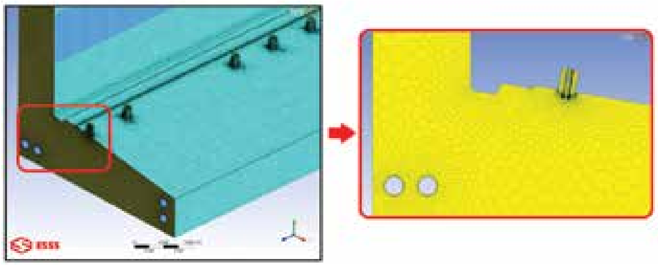


Fig. 3 - Mesh applied in all cases

Fig. 2). Four cases were simulated, two with 6 burners and two with 9 burners. In all the cases simulated a 1.5 million mesh with hexahedral cells was used in the porous region, whilst for the rest of the domain polyhedral elements were applied (Fig. 3).
The study was developed in ANSYS Fluent in a steady state regime. The combustion was evaluated with Eddy Dissipation Model (EDM) which accounts the effects of mixing degree in chemistry due to turbulence using 2 step methane-air kinetic. For turbulence an k-ω SST model was used and for radiation a Discrete Ordinates (DO) model. An additional sub-model was applied, the Weighted Sum of Grey Gases (WSGGS) model, to consider the participation of gray gases in radiative heat transfer, and the buoyancy effects were accounted for in all simulations.

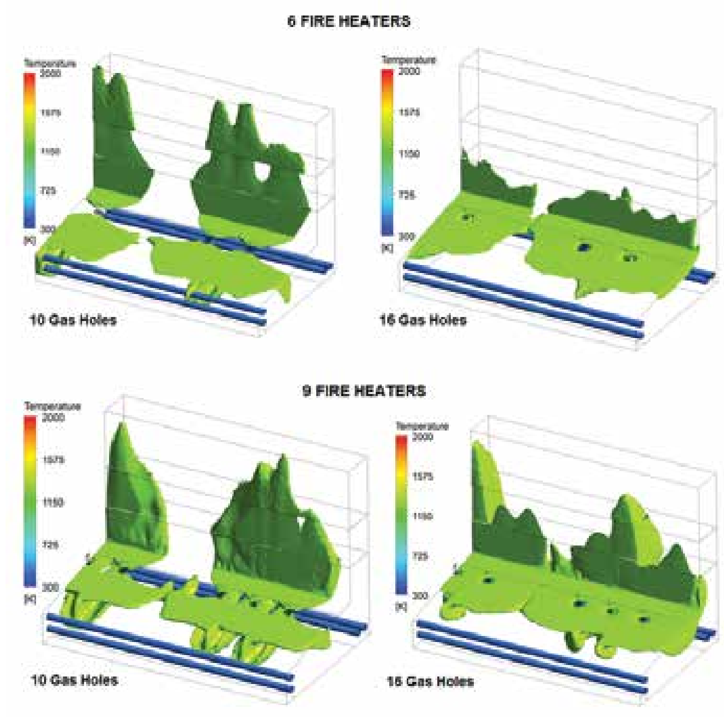


Fig. 6 - Iso-surface of temperature, 1250 K

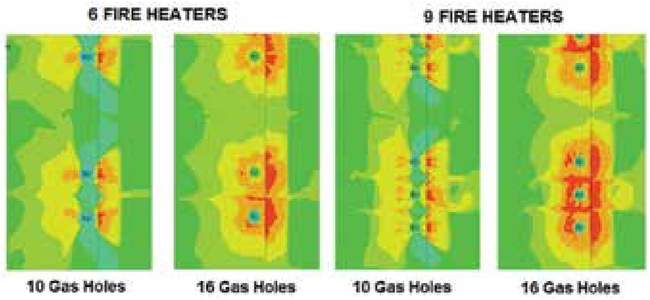


Fig. 4 - Temperature contour on the plane parallel with the ceiling

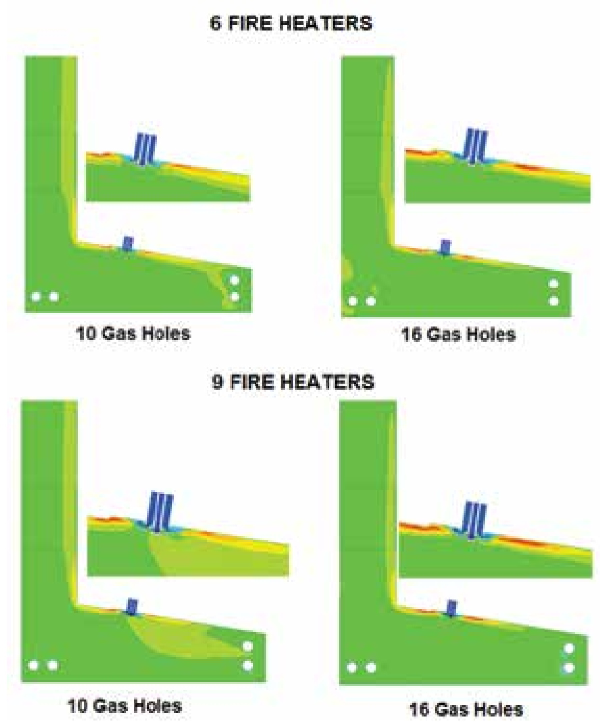


Fig. 5 -Temperature contour on the plane that cuts the fire heater

The main objective of the CFD study was to compute the dynamical and thermal behaviours of the fire heaters, since it is important that the flame doesn't lean the tubes, avoiding region with high heat flux on the tube walls, which can cause some problems such as coke formation and, hence, flow obstruction. Therefore temperature contours in two different positions, parallel with the ceiling and at a plane cutting the fire heater, were used to evaluate the results with iso-surfaces of temperature (1250 K) to evaluate the flame position.
As seen in figures 4, 5 and 6 the cases with six fire heaters and 16 gas holes presented better results regarding the tubes in convection. These results clearly show how powerful a tool ANSYS FLUENT can be during the designing phase of equipment as complex as furnaces, allowing the operator major savings in money and time.

Raphael Bacchi, Ana Faria
ESSS – Engineering Simulation and Scientific Software
Ricardo Serfaty
Petrobras

ROCKY DEM and ANSYS CFD COUPLING: a powerful tool for simulating granular-fluid systems

The coupled DEM (Discrete Element Method) – CFD (Computational Fluid Dynamics) approach is a promising alternative for modeling granular-fluid systems, enlarging the range of coupled particle-fluid processes that can be managed with numerical simulations. Complex phenomena such as pneumatic conveying, granular drying, slurry flow inside grinding mills, or even chemical reactions between particles and fluids can be handled with these powerful tools.

A few successful coupling cases using DEM and CFD packages were published, along with experimental validation, which indicated the feasibility of this modeling approach to take into account both the effects of fluid volume displacement by particles as well as the drag force of the .

Currently, there are two ways to couple DEM and CFD technologies: one-way coupling, in which only the fluid flow affects the particle movement, and two-way coupling, in which the particle flow also influences the continuous phase behavior.

However, considering the industry necessities, there is a strong desire to have an adequate results-guaranteed coupling. This is exactly why ESSS, ANSYS channel partner in Brazil, GDI, creator of the original ROCKY software, and ANSYS itself decided to combine their efforts and develop 1-way and 2-way coupling between ROCKY and FLUENT.

ROCKY is a powerful DEM software package, capable of performing 3D simulations of granular flow through a handling system. It uses real particle geometries (not a combination of spherical particles, for example), and simulates and analyzes aspects like 3D surface wear modification, particle breakage, sticky particles, and rotating and vibrating boundaries. For the past several releases, ROCKY has been coupled with Finite Element Analysis (FEA) ANSYS software, allowing engineers to evaluate the tensions stresses and forces generated by these particles as they interact with the handling equipment, such as chutes and conveyor belts.

ROCKY can run in parallel, under a CPU or, even better, a GPU cluster system. ANSYS FLUENT package is one of the world leaders for CFD applications, and its HPC CPU capability is very well documented. Both ROCKY and FLUENT have been extensively validated and tested.

Considering all these aspects, the work that ESSS, GDI and ANSYS accomplished now allows FLUENT to work with ROCKY in parallel CPU and GPU capabilities, which leads to a very trustworthy, robust, and fast DEM-CFD simulator. In addition, this parallel processing methodology opens up the possibility of simulating large multiphase flow devices, with many millions of different sized and shaped particles, which is historically a weak spot with DEM technology.

This integration is deeply rooted in both software packages. The interface inside FLUENT makes it easy to start this multiphase coupling with ROCKY (Figure 1). After a few simple steps, both solvers are set to work in parallel.

To show the coupling capabilities of ROCKY and FLUENT, some examples are presented below. A windshifter, equipment typically used in industrial waste processes to separate light from heavy particles, was modeled using both a pure DEM approach and a one-way coupling approach

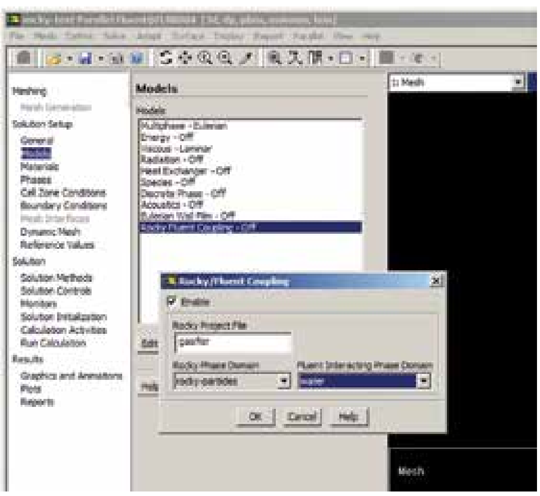


Figure 1. ROCKY-FLUENT setup panel inside of the Models panel of FLUENT. The interface provides an easy setup for two-way coupling simulations

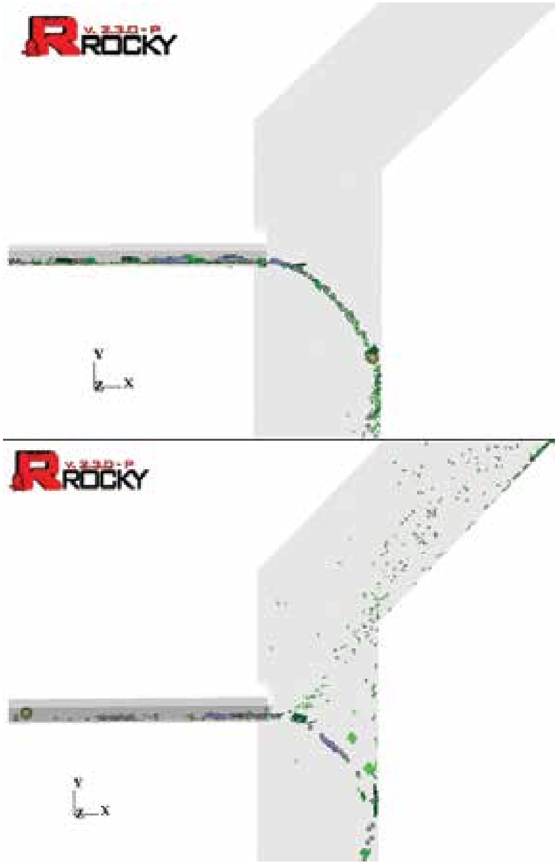


Figure 2. Comparison between pure DEM (above) and one-way CFD-DEM approaches (below) for a windshifter case

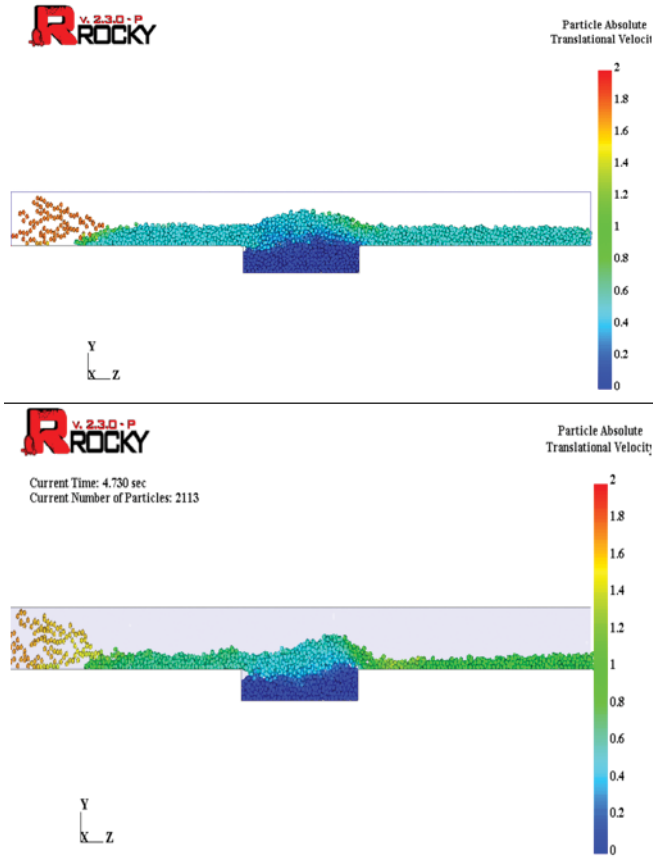


Figure 3. Comparison between one-way (above) and two-way (below) CFD-DEM approaches for a cavity filling case

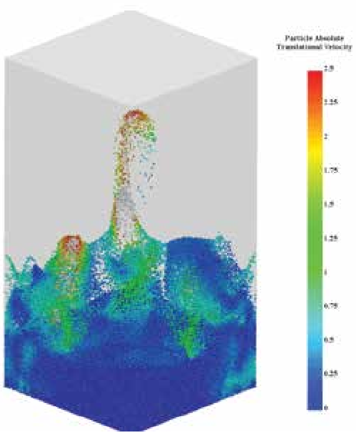
between ROCKY and FLUENT. Particles were dropped into the shaft and, depending on their sizes, shapes, and densities, they went up (light fraction) or down (heavy fraction). The comparison in Figure 2 shows the differences between the two results and the possibility of investigating the influence of wind speed and feed rate on separation efficiency.

For comparing one-way and two-way DEM-CFD approaches, a simple cavity filling case was simulated and the simulation results for both approaches were compared (Figure 3). In the two-way case, the formation of a wave pattern of solids above the cavity can be observed.

A more complex case is shown in Figure 4. This case is the simulation of water injection (in three injection points) below a bed of initially still particles. The image at the top shows the ROCKY results for ~2s of physical simulation time. At this point of the simulation, the particle bed is entirely agitated and some rapid streams of particles are starting to be dragged as the fluid finds shortcuts and accelerates among the particles. The images at the bottom of Figure 4 show the results of water velocity from the FLUENT solution, and also shows the iso-surfaces of the particles volume fraction that have been imported from the ROCKY solution.

As can be seen in these examples, ROCKY- FLUENT coupling is a powerful tool for designing and troubleshooting particulate processes using simulation technology, enabling engineers to analyze a large range of processes in many different industries such as oil and gas (drilling, for example), agroindustry (grain transportation and drying, for example), pharmaceutic (transport), mining (mineral washing, for example), and many others.

João Américo Aguirre Oliveira Jr., Lucilla Coelho de Almeida, Clarissa Bergman Fonte
ESSS – Engineering Simulation and Scientific Software
Alexander Potapov
GDI – Granular Dynamics International



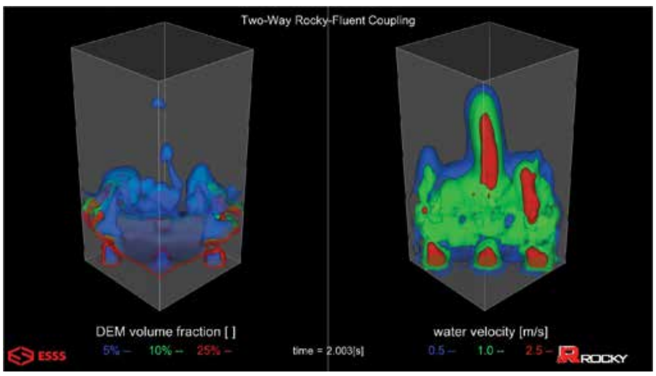


Figure 4. Two-way coupling simulation of injection of water below a bed of particles, ~400,000 particles. Images show ROCKY results (top) and FLUENT results (bottom) after ~2s of physical time

CAE applications for Valves in the Oil&Gas sector

Valves are mechanical components used in all engineering sectors. They are particularly essential components of plants in the Oil&Gas, petrochemical and energy sectors. Their aim is to intercept and to control the flow of potentially dangerous fluids and they are often located in ambient and operative critical conditions. For this reason valves manufactures have to design them guaranteeing a high degree of reliability and durability, with the minimal possibility of required maintenance. The capability to guarantee a high level of quality is therefore a competitive advantage.
Virtual prototyping tools (CAE) significantly contribute to increasing this factor. There are a lot of CAE applications involving both structural aspects related to the fatigue, which are able to correct the functioning and the durability of the product, through fluid-dynamics optimisation of dissipation and noise. Above all, the structural analysis enables an accurate calculation of dimension for valve bodies and covers, respecting regulation requirements and the operative conditions that are often led by internal pressure loads and thermal coactions. It's important to highlight that the optimized dimensioning enabling the ability to reduce consistently the weight of a valve, even in valves of a large scale, more than what would be possible from the conservative dimensioning given by a "traditional" conventional approach. Benefits of a valves reduction in weight involve both direct materials costs and indirect costs such as transport and installation.

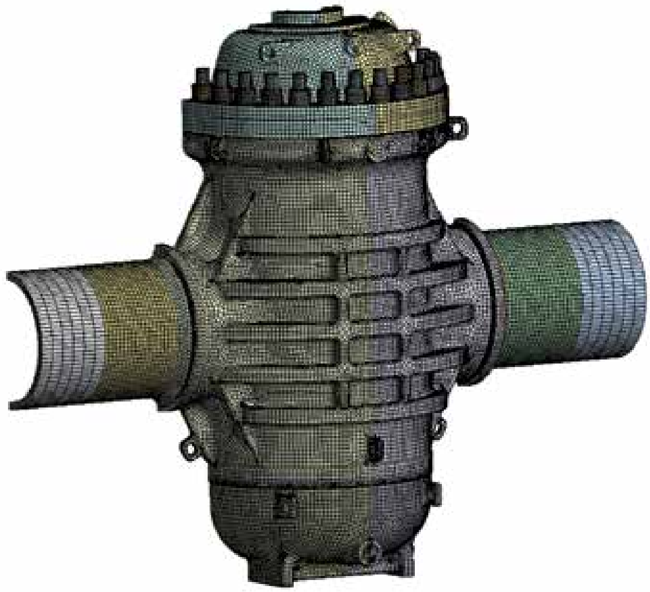


Fig. 2 – Gate valve mesh

Fig. 1 – Ball and socket contact simulation

Benefits of making simulations directly on the CAD geometry are high. This method will allow for geometric simplifications and reduced modelling hypothesis presenting more accurate results and forecasts. Coupling the "overall" evaluations shown above with the numeric techniques, it's possible to detect accurately the "local" aspects equally important to guarantee the success of design. A fundamental aspect related to the product purpose, is the verification seal; the most traditional example is the ball and socket coupling dimensioning in which it is possible to refine the sockets

profile and consequently their stiffness, in order to guarantee an adequate contact and reaction condition to the seal.
In this type of simulations it is fundamental to accede to complex simulation algorithms, complex because they have different non linear aspects. These algorithms have to be robust and efficient to maintain simulation times compatible with the timing of design. Other aspects of validation include, deformation constraints in place, available tolerancing, tightening optimisation of flanged couplings, check of dimensioning in organs connected to execution mechanisms that have to transfer manoeuvre couples.
Fluid-dynamics analyses (CFD – Computational Fluid Dynamics) are used in several applications: the most common ones concern evaluations of performance parameters such as pressure fall and the pressure coefficient cv.

Other simulations are related to layout optimisation in turbulence phenomena and to the generated noise, or cavitations, or to the search of the equilibrium position of organs under fluid flow action (i.e. swing check valves). In these cases simulations gather further advantages interacting with the structural calculation (FSI – Fluid Structure Interaction) enabling dynamic phenomena analysis such as an impulsive closure due to the line breakage.
Numerical techniques effectiveness has recently increased thanks to the high impulse received by the availability of work places which combine operational eases and extreme numeric efficiency (i.e. availability of parallel and high-performance solvers).

On the above mentioned applications EnginSoft developed considerable engineering and technology transfer activities towards several Italian producers. EnginSoft, thanks to the pluri-annual experience, aims to be the partner of all companies willing to improve their design process, and to implement the virtual prototyping. EnginSoft provides, in addition to the engineering activities, leading industry technology and training on simulation technology transfer activities.

Sergio Sarti - EnginSoft

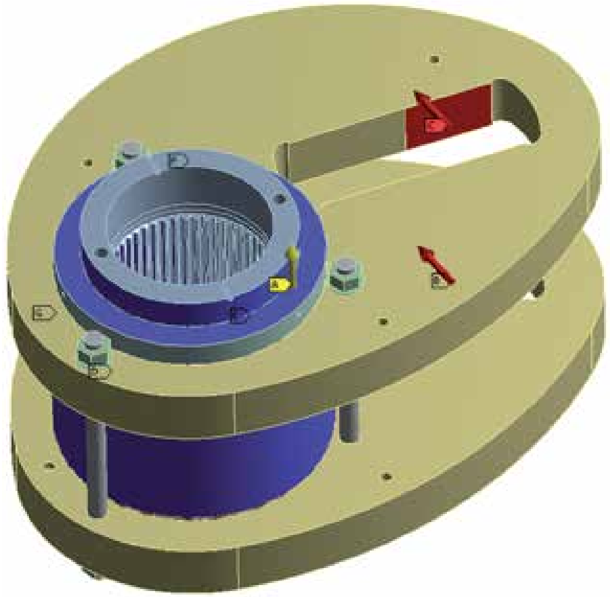


Fig. 3 – Glifo FEA model for a valve actuator

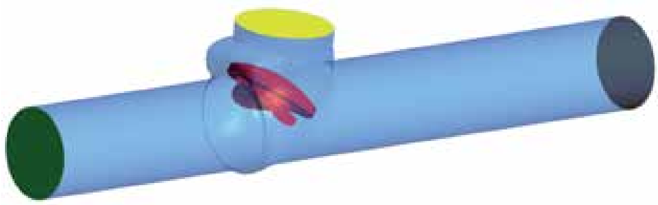


Fig. 4 – CFD analysis of a shutter for control valves

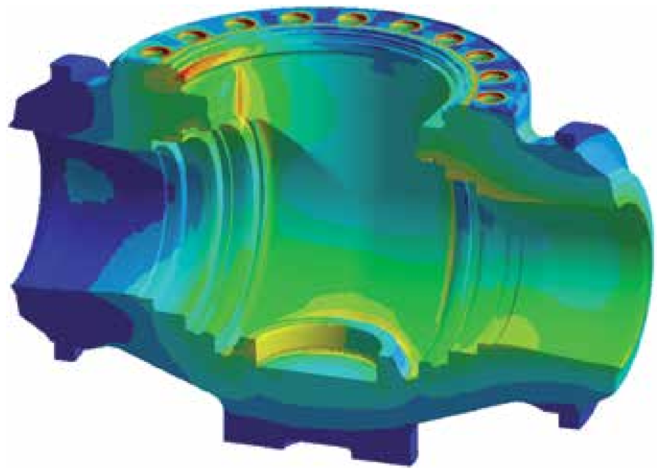


Fig. 6 – Stresses calculated on the body of a top-entry valve – FEM analysis

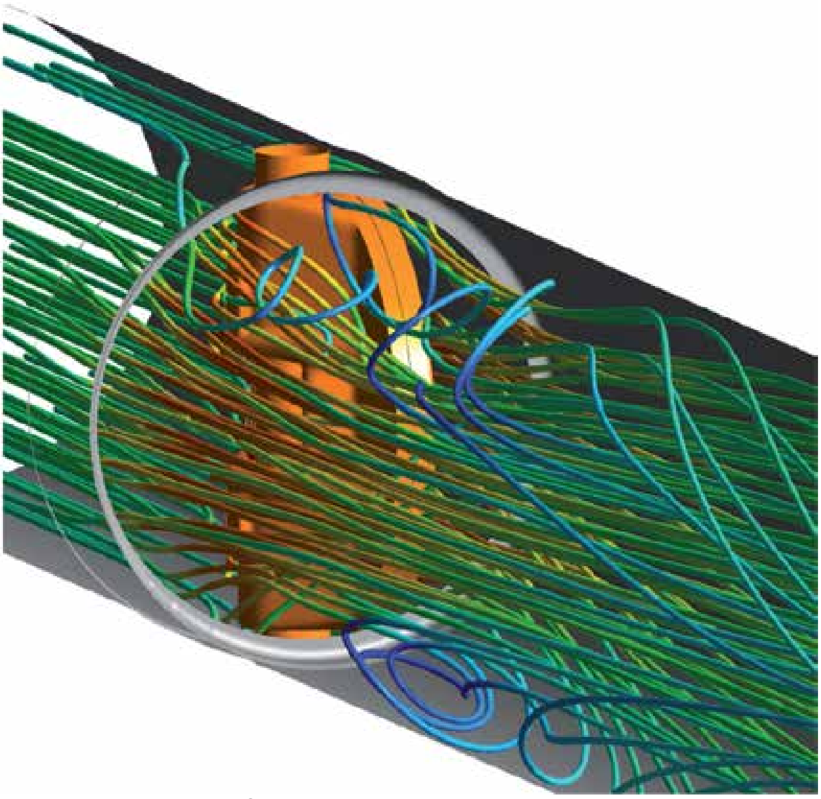


Fig. 5 – Flow lines simulation – CFD Analysis

Second HYDRO POWER PLANT Turkey is taking off

High quality engineering speaks Italian

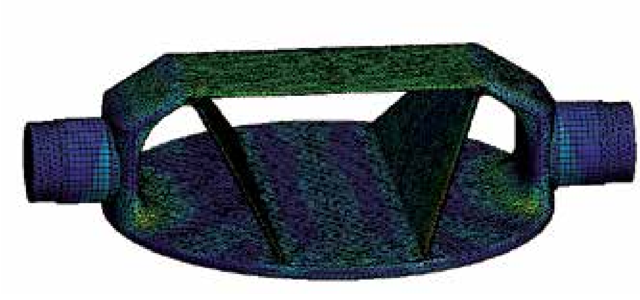
The Kargi Project that consists of the construction of the second major hydroelectric power station in Turkey, will be completed in the second half of 2013 with the direct involvement of expertise, resources and technologies coming from all over Europe.
The power station located about 200 km northeast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

CIB S.R.L. CARPENTERIA INDUSTRIALE BRESCIANA

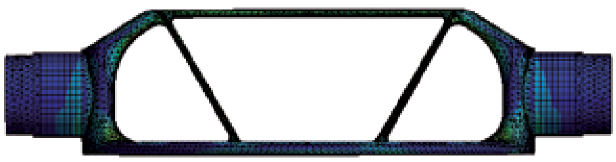
These have allowed the designers to develop the butterfly-shaped cut-off profiles according to the hydrodynamic characteristics that are adequate to reduce the load loss coefficient to a minimum extent. The profile morphology, particularly enhanced in order to meet the project specifications, required targeted interventions to achieve shape optimization and, as a consequence, to obtain a real hydro-dynamic and highly efficient profile out of a rough manufactured component.
The fluid-dynamic behavior has been simulated for different partitions of the cut-off. Two of them proved to be particularly relevant and worth of an in-depth description:

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

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



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



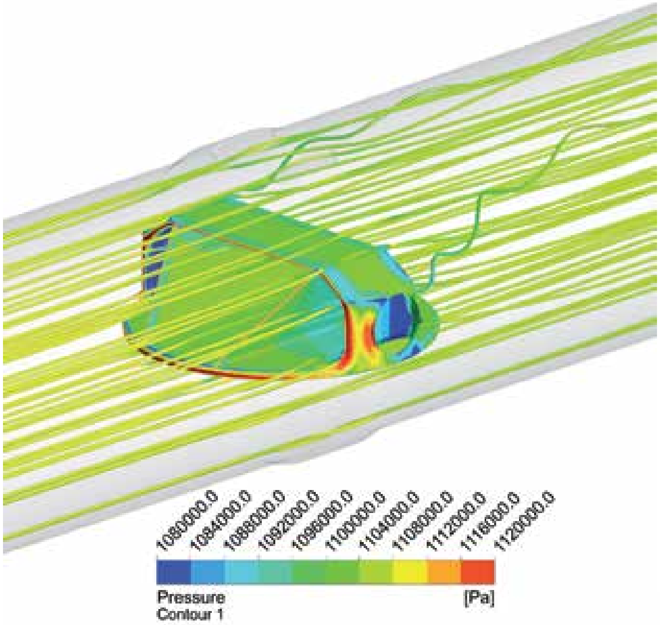
this purpose. Practically speaking, the test case consisted of a huge bell, flanged to the valve, in which water was injected and artificially led to a test pressure through pumps and load control systems. In addition to the checks for tightness, the inspectors from Voith have also experimentally detected the correlation between the experimental and the theoretical distortions. This has been done using simulation and through constantly monitoring the measurement systems throughout the test.

So what can we say? …That high quality engineering speaks Italian, also in Turkey.

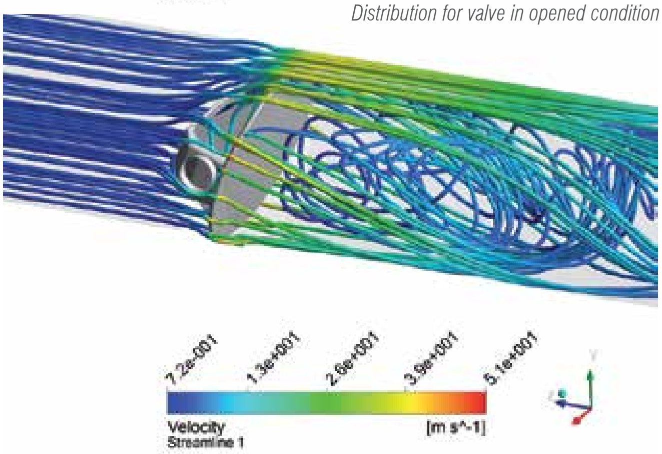
The team of engineers that have developed the Kargi Project will be present at the EnginSoft booth at the OMC 2013 – Oil&Gas, Offshore and Inshore Exhibition, scheduled March 20 – 22 in Ravenna, Italy. We look forward to meeting our customers and friends who want to hear more about this and other completed projects!

Livio Furlan - EnginSoft

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



Distribution for valve in opened condition



Francis Turbines

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

As a compensation, Francis turbines have the best efficiency peak of all turbines, which when used for huge plants, can reach a range of 94% and above.

CO_2 Stripping Column Effluent System Application employing Saipem mechanical innovative Technology

Paolo Minola, Saipem S.p.A. San Donato Milanese, Italy

In this project, we present the design of two effluent piping systems for use in CO2 stripper columns in Ammonia plants. The main issues addressed include a complex lay-out due to the double tangential feed inlet and the presence of a bi-phase liquid, which may lead to vibrational phenomena induced by fluid dynamic instabilities. The effluent system presented features a number of design enhancements with respect to previous designs. Its geometric form and the use of a Y-shaped fitting eliminate fluid dynamic issues caused by the presence of a bi-phase mixture, while a system of restraints resolves issues of a mechanical-structural nature.

The specific design context in fact led to a number of targeted and in certain cases innovative solutions in terms of both design and construction, including the system's overall mechanical design, the construction of the Y-shaped fitting, which employs heat treatment techniques and NDT methods, the system's complex spatial configuration and stringent fabrication and assembly tolerances. Furthermore, the design of the system of compact elastic restraints and special non-welded supports to protect the elbows of the effluent system provide it with a high degree of structural reliability, particularly considering the structure's overall height of approximately 40 meters.

1. INTRODUCTION

The objective of this article is to describe the Y-Tee innovative device, which Saipem implemented for the effluent system connected to the High and Low pressure ammonia flash Drums. The use of Y-Tee and other integrated fittings such as fluid rectifiers prevented the occurrence of typical problems of these types of circuits concerning vibration phenomena, complexity of the geometrical lay-out and criticality of the assembly phase. Saipem improvements led to the avoidance of fluid-dynamic instabilities as the internal cross separator of the Y-Tee devices kept a balanced division of the fluid path downstream, thereby reducing turbulence and breaking up bubbles. Their particular shape allowed for an optimized circuit lay-out, starting from the preliminary definition of the outlet piping angles. Y-Tees were also used as templates during the erection phase, thus facilitating adjustments in the field. Also, complementary components such as fluid rectifiers helped to avoid twirling phenomena in the fluid moving upwards, thus preventing the occurrence of vibrations and keeping losses within defined limits. Furthermore, the choice of the type of compact elastic restraints and the design of special non-welded supports to protect elbows ensured a high degree of structural strength.

The work-flow process implemented by Saipem will be adopted as reference guideline for incoming projects as it

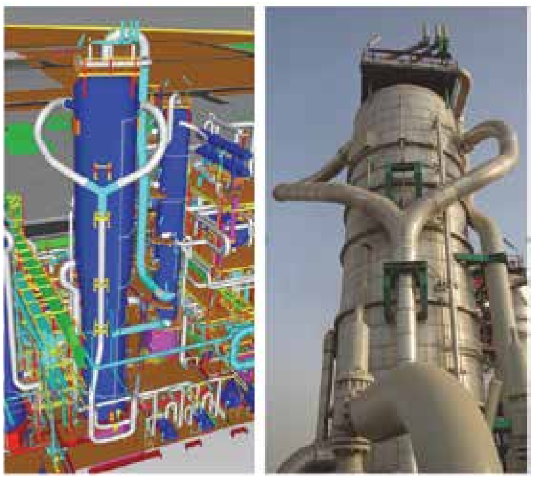


Fig. 1 - Ammonia flash Drum effluent system

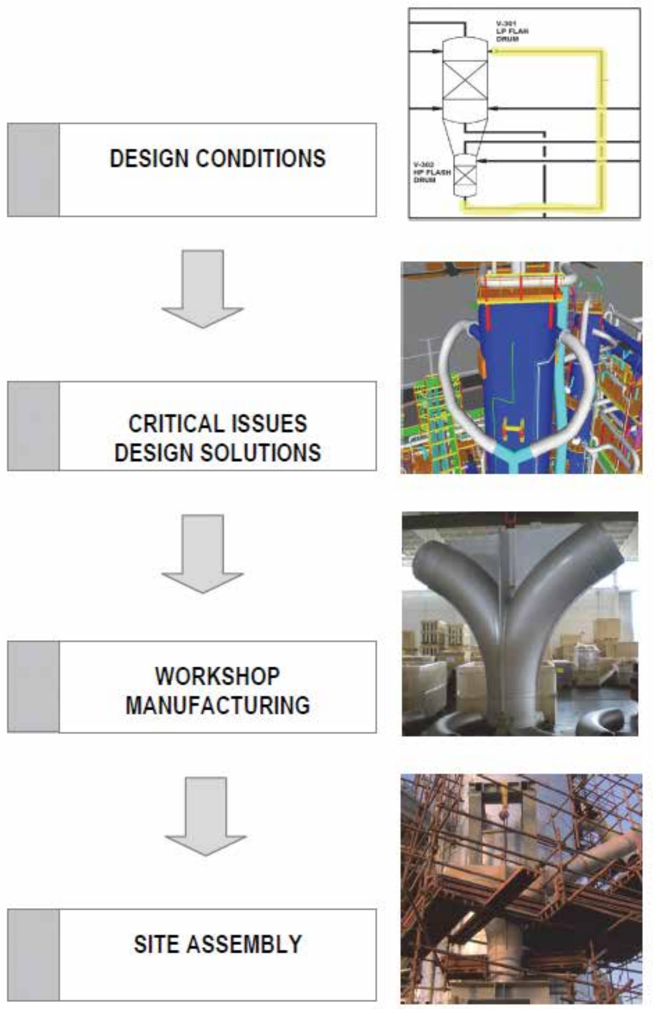
integrated the fluid-dynamic and the mechanical aspects. The designed fittings constituted innovative technological solutions that enabled to gain a high level of plant reliability: positive expectations concerning the benefits gained by their insertion on ammonia-urea plants have been fully satisfied. Saipem accomplished this goal through a multidisciplinary approach made possible by the collaboration between specialists in the Saipem headquarters, on site and by a skilled team in the workshop. The team applied Saipem's instructions step by step with respect to both the design and the construction aspects of the project.

The entire design of the ammonia effluent lines was implemented in the following Ammonia-Urea plants that have been in operation for several years with no evidence of fluid-dynamic instabilities due to unbalanced forces:

- ENGRO DAHARKI COMPLEX - Pakistan
- QAFCO5 FERTILIZER COMPLEX – Qatar

2. FLOW-CHART

The work flow includes the following steps.



The following paragraphs will outline the main aspects of each work phase.

3. DESIGN CONDITIONS

The effluent circuit is part of an ammonia plant that is based on Haldor Topsøe's technology. The main process steps necessary for the production of ammonia are listed below:

- The desulphurized hydrocarbon feed is reformed with steam and air into raw synthesis gas (process gas). The gas contains mainly hydrogen, nitrogen, carbon monoxide, carbon dioxide and steam.
- In the gas purification section, the CO is first converted into CO2. Then the CO2 is removed from the process gas in the CO2 removal section.
- The purified synthesis gas is compressed and then routed to the ammonia synthesis loop, where it is converted into ammonia.

The piping system is part of the gas purification section and operates with BASF solvent aMDEA®. The aMDEA solution contains 40 wt% aMDEA; an activator increases the mass transfer rate of CO2 from the gas phase to the liquid phase. The rest of the solution is water. The system is outlined in the flow diagram: the effluent considered circuit is highlighted in yellow.

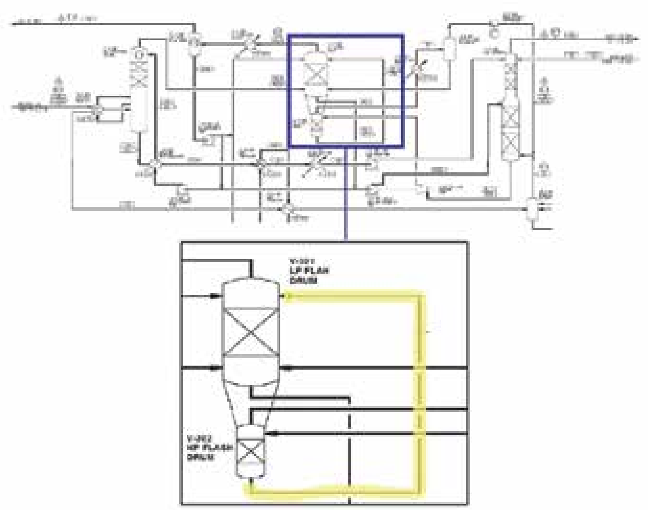


Fig. 2 - Process flow diagram of gas purification section

The process cycle starts in the HP flash drum (V-302, 1st stage), where most of the dissolved inert components are released. The rich solution continues to the LP flash drum (V-301,2nd stage), where most of the CO2 is released from the solution at a defined value of pressure. Level control valves LV-3011A/B that act on the level in the vessel V-302 were positioned at the bottom of the circuit. Their function is to ensure sufficient hydrostatic head and avoid excessive flash at the valve outlet. Downstream of the valves the piping lay-out presents a vertical development of about 20m after which the line is split into two branches by the insertion of a Y-Tee. The solution is fed to the vessel V-301 by means of two tangential inlet nozzles.

The percentage of released vapor increases along the rich solution line. Small gas bubbles dissolved uniformly in the rich solution inside the piping should not cause any problems. On the other hand, as large gas bubbles can cause vibration, the circuit shall be properly designed and strongly supported. Regarding the geometry of the system, the total difference of level between the bottom and the top of the column is about 40m. The types of loads acting on the circuit are of both static and dynamic nature. Static loads are constituted by the dead load of the piping, fittings, and internal fluid plus the pressure and temperature of the fluid. It should be noted that most of the static load is distributed over the vertical part; moreover, the supporting bracket structures are welded on the Vessel shell.

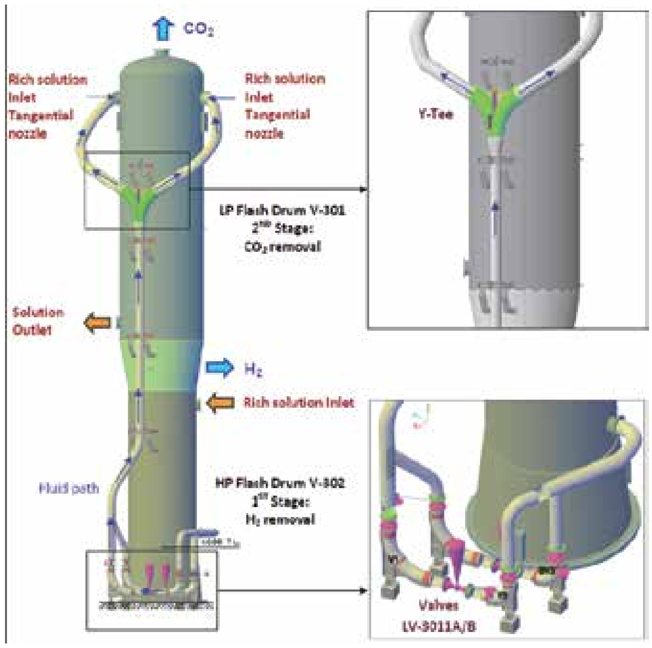


Fig. 3 - Overview of the system

Dynamic Loads are not defined univocally as they can be significantly reduced or eliminated if appropriate design choices are made. A list of the main load cases follows. The design was intended to keep the effects on the system within acceptable limits.

- Slug Flow with consequent vibrations may occur if vaporization takes place in the vertical piping instead of inside the column.
- An additional dynamic load can arise from losses of head due to abrupt directional changes of the fluid path.
- Another issue are fluid dynamic instabilities due to the presence of unbalanced forces on the column inlet nozzles that could lead to flexural and torsional oscillation phenomena.

4. CRITICAL ISSUES AND DESIGN SOLUTIONS

Having to meet both process and mechanical requirements led to the use of targeted solutions for specific problems. The mechanical design was intended to ensure acceptable levels of stress on all the components of the cinematic chain including elbows and nozzles. The added improvements provided for critical items regarding the design, workshop and assembly phases were aimed at obtaining a high quality level also through innovative technological components.
The main critical issues and the solutions applied to ensure the system operate in a safe conditions are described in detail in the following paragraphs.

4.1 Process and Mechanical Requirements

A requirement was that the liquid phase of the fluid rich in CO2 shall not vaporize in the vertical part of the circuit. If this phenomenon occurs a slug flow condition with induced vibrations will be generated. A slug flow condition on the two branches downstream the Y-Tee and unbalanced dynamic forces of the tangential nozzles shall also be avoided. Appropriate mechanical solutions concerning both the asset of the whole circuit and the Y-Tee were provided. The goal was to ensure that the fluid is separated uniformly and head losses are minimized in order to prevent vibration phenomena. The implemented solutions consisted in the insertion of flow rectifiers on specific sections of the piping and on the Y-Tee. An ad hoc component for the Y-Tee was also designed.

4.1.1 Flow Rectifiers

A Flow Rectifier device was inserted in the piping at the elevation of 9535 mm and in correspondence of the Y-Tee section, as indicated in the figure below.

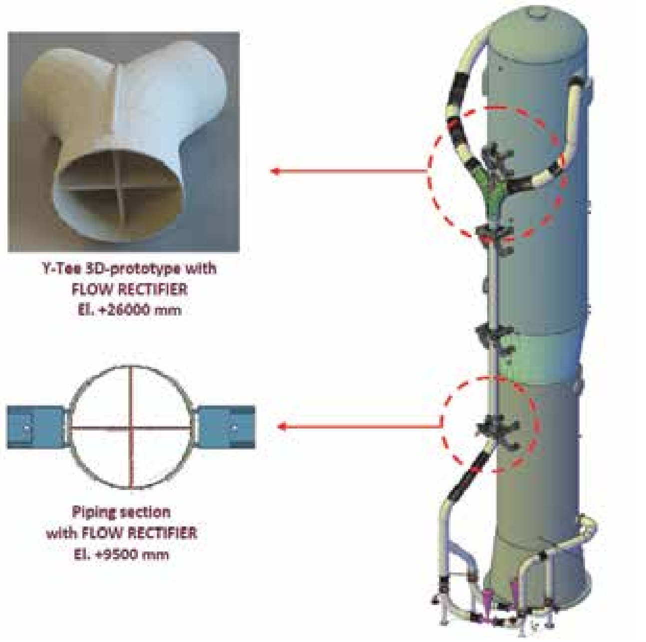


Fig. 4 - Flow Rectifiers

It consists of a cross that forces the fluid path to keep a straight direction thus avoiding turbulence and the detachment of the gas phase. It also has a mechanical function as it strengthens the piping section.

4.1.2 Y-Tee

The main requirements were to balance the two branches downstream of the split section thus avoiding fluid-dynamic unbalanced forces acting on the two tangential nozzles. The Y-Tee should also uniformly separate the fluid. It was necessary that the length of the piping portion downstream of the Y-Tee be minimized in order to reduce fluid losses.
The Y-Tee makes it possible to uniformly separate fluids through a system of cross baffles that are part of the device. The internal baffles channel the process fluid in the two outlet sections improving fluid-dynamic efficiency. A balanced profile prevents vortex formations and turbulence in the fluid, which are the main cause of vibration in the connected piping.

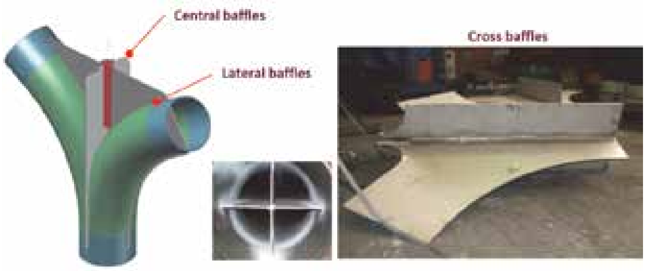


Fig. 5 - Y-Tee

These baffles give the object high mechanical resistance thus preventing ovalization of the sections and local buckling. Furthermore, the presence of a connection for the lifting lug makes the Y-Tee a self-bearing structure that also acts as a support to the inlets and outlets. The new fitting (Y-Tee) made it possible to optimize the layout of the entire circuit thus reducing the number of bends and the total length, and eliminating abrupt changes of directions. A schematic of the initial and of the optimized lay-out carried out after the design of the Y-Tee is shown in the figure below.

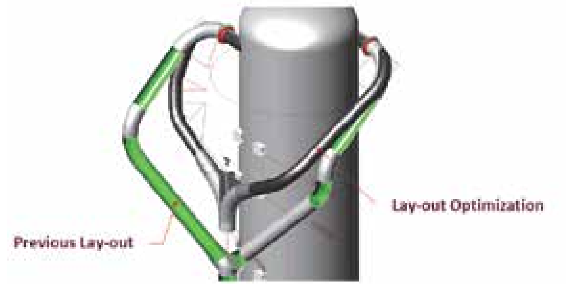


Fig. 6 - Comparison between the initial and the optimized lay-out

The Y-Tee discussed here was analysed with a computational fluid dynamics (CFD) approach whose goal was to balance the load at the top of the column. The results helped pinpoint potential critical conditions and indicated which were the most fitting implementations. The mechanical solutions were carried out on the basis of the fluid dynamic analysis and gave structural integrity to the component. Once the design of the circuit and of the single components was completed, a mechanical (FEM) analysis was also performed to ensure the stress distribution was within the allowable limits. A general overview of the performed CFD and FEM analyses follows.

4.2 CFD Analysis

Computational fluid dynamics (CFD) analysis was necessary to evaluate critical fluid dynamic regimes and the flow field inside the pipe in order to avoid harmful vibration. Under normal operating conditions, a multiphase flow composed of amine solution and gas is present inside the pipe. Different flow regimes were took into account: flushing, start-up, normal and shut-down. The CFD analysis was performed on the whole circuit including the Y bifurcation. The objective was to evaluate critical fluid dynamic regimes and the flow field inside the pipe.

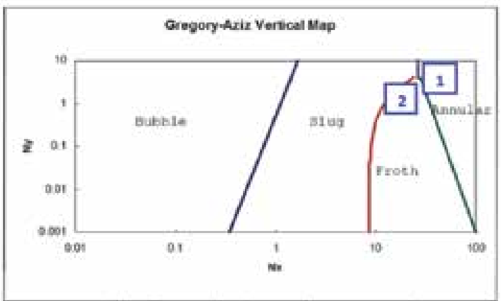
4.2.1 CFD Model

Two principal flow regimes are applied to the system: single phase liquid flow at high pressure level (flushing, start-up and shut-down) and multiphase flow (liquid + gas) at low pressure value (slug, froth).

The CFD analysis was set up for two extreme scenarios:

- Single phase flow with the maximum value of flow rate and density;
- Multiphase flow with the maximum values of gas flow rate and gas volume fraction.

All CFD analyses were computed assuming a single phase fluid with average flow and density properties. In particular, for the simulation of the single phase flow, an incompressible liquid was described, whereas an equivalent gas was assumed for the multiphase flow analysis. The circuit is characterized by two different flow regimes, localized in two areas of the Gregory-Aziz Vertical Map determined by liquid and gas velocity; Gregory-Aziz maps identify the flow pattern: the more critical fluid dynamic phase has been identified with the Normal phase (multiphase flow with maximum values of flow rates). The geometry was built using ANSYS Workbench meshing tools for CFD. For each run a stationary flow was defined in turbulent regime. A fixed flow rate at the inlet section and constant values of both temperature and pressure were chosen as boundary conditions.



	Flow Rate [kg/h]		Density [kg/m³]		Velocity [m/s]		Volume Fraction	
1 Normal phase	Liquid	Gas	Liquid	Gas	Liquid	Gas	Liquid	Gas
	3455912	79496	1060	2.18	3.236	36.199	0.082	0.918
2 Normal phase	Liquid	Gas	Liquid	Gas	Liquid	Gas	Liquid	Gas
	3455912	79496	1060	2.18	1.436	16.088	0.082	0.918

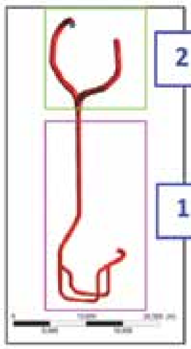


Fig. 7 - Gregory-Aziz Map

Starting from average information of the flow, it is possible to estimate the oscillations of the device hypothesizing a regular vortex detachment from the internal Y partition wall. The oscillation frequency can be obtained from an experimental chart that relates the Strouhal and the Reynolds numbers. In the analysed case, a Strouhal number equal to 0,24 corresponds to an oscillation frequency of 330 Hz.

4.2.2 CFD Results

At the end of the computation, results were used to extract significant virtual measurements and later used to rate the effectiveness of the analysed geometry. The pressure drop between inlet and outlet section was calculated, weighting the contribution of each geometrical singularity along the path. Levels of pressure throughout the pipe, fluid velocity and forces were extracted as well.

Representation of the streamlines has allowed highlighting some vortices in the flow within the duct. Those vortices suggested the need to install a cross-flow baffle in the piping downstream of the Y bifurcation. The most critical situations can be simplified considering two extreme scenarios for a CFD analysis:

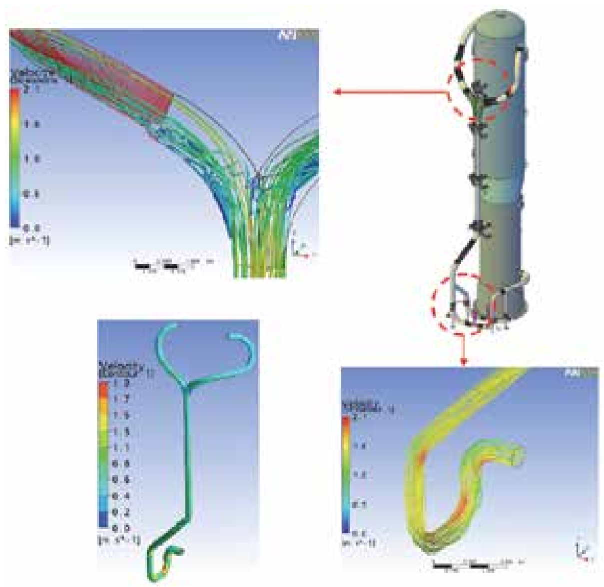


Fig. 8 - Rotational flow inside the pipe represented by streamlines

- single phase flow considering liquid flow
- single phase flow considering gas flow

The pressure drop between inlet and outlet section was calculated, weighting the contribution of each geometrical singularity along the path. Levels of pressure throughout the pipe, fluid velocity and forces were extracted as well. CFD analyses, in both scenarios, show important vortex core regions (rotational flow) inside the pipe, in particular before the partition wall:

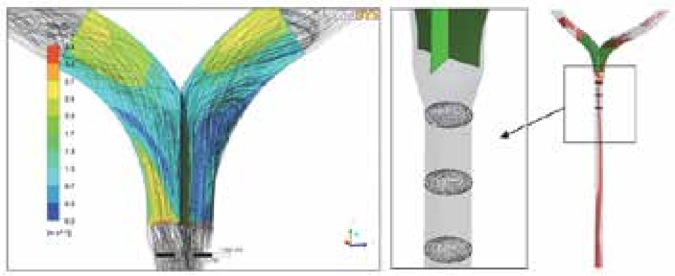


Fig. 9 - Vortex core region before Y-Tee

A negative effect of this interaction is also a non-uniform distribution of flow between the two outlets, in particular considering the liquid analysis:

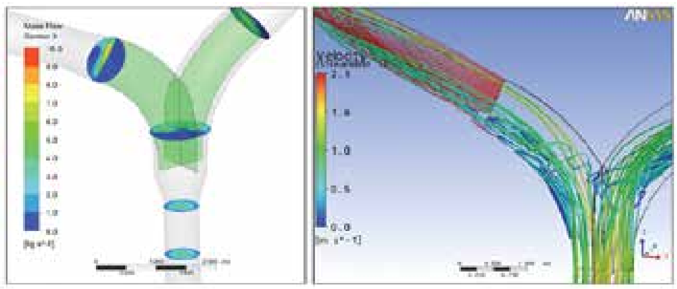


Fig. 10 - Flow-distribution

An internal device (flow rectifier) was inserted upstream of the Y-Tee to mitigate the effect of the vortex core regions. A modal analysis was also performed in order to evaluate possible resonance phenomena due to the coupling between the modal frequencies of the structure and the vibration frequencies induced by the flow. No resonance was detected.

4.3 FEM Analysis

Numerical analyses were all performed within the ANSYS Workbench platform, which allows managing both CFD and FEM problems in a similar manner. The ANSYS Workbench easily allows passing information from one environment to the other one, giving the possibility to realistically model the true fluid-structure interaction. Finite element method (FEM) analysis was applied to evaluate the stress state under several design conditions for pipes, bends, supports, welds, flanges and bolts. For the structural verification of the piping, represented as a pressure component for calculation purposes, ASME code for boiler and pressure vessel were used as a reference for the analysis, while for support items AISC code was considered. Computational fluid dynamic (CFD) analysis was necessary to evaluate critical fluid dynamic regimes and the flow field inside the pipe in order to avoid harmful vibration. Under normal operating conditions, a multiphase flow composed of amine solution and gas is present inside the pipe. Different flow regimes were took into account: flushing, start-up, normal and shut down. The analyses played an invaluable role in the proper development and testing of the new concept of a piping system with a Y bifurcation. This was also true for the support configuration and the connection equipment. The other objective was to eliminate risks due to harmful vibration of the structure caused by fluid regimes inside the pipe.

4.3.1 FEM Model

The objective of the FEM activity was to evaluate the behaviour and stress state of both the Y-Tee pipe and the reinforcement plates. The parts calculated and verified were:

- Stress on the pipe, checked with reference to Allowable Displacement Stress Range reported in ASME B31.3, and assuming a number of 7000 working cycles during the expected service life of the piping system.
- Stress on Y-Tee supporting structure. Here the admissible stresses are in relation with yield and ultimate strength of the support material, as specified by AISC rules.
- Welds, with reference to the seam weld direction, comply with the stress limits for both the normal stress and the shear parallel stress, as specified by AISC rules for welds.
- End flanges stress verification, with reference to admissible value reported in both ASME B31.3 and ASME VIII div.2 appendix 4, as specified in par. 4.1 for pipeline.

Bolt pre-load required for gasket seating, according to ASME B31.3.
Several load cases were considered for the analysis: hydraulic test, pipeline installation, chemical cleaning start up, normal run. Each load case is characterized by pipe internal pressure, temperature of flowing fluid, and temperature of supporting column (either linearly variable or constant). In addition, loads due to pipe self-weight and hydrostatic pressure were also introduced.

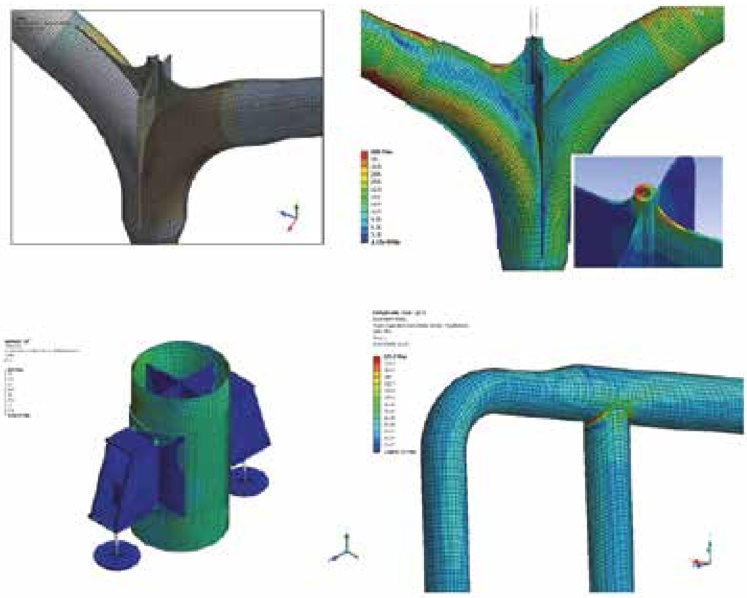


Fig. 11 - MESH Images

4.3.2 FEM results

The evaluated stresses along the pipeline are below the admissible stress limits (Fig. 13). Locally, in correspondence of the bypass pipes, a more accurate mesh was modelled using solid sub-models. Eventually every pipe check was satisfactory from the strength resistance point of view. Similarly, all flanged connections were successfully verified. Bolts pre-load required for gasket seating was calculated according to ASME B31.3.

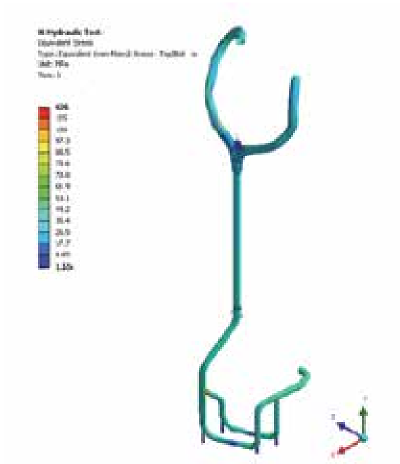


Fig. 12 - Piping equivalent stress intensity

4.4 Constraints

Supports for static loads characterized by innovative mechanical contents were inserted in the most critical points in order to ensure the structural integrity of the circuit. The main requirements set out after a general overview of the system from a mechanical point of view concerned the following issues:

- Minimize the weight of the supports, especially those of the spring hangers that bear the vertical loads. Load concentration shall be avoided in order to preserve the Vessel shell to which they are welded;
- Avoid stress concentration by distributing the applied loads and minimizing the use of welded components.

The solutions implemented are described in the following paragraphs.

4.4.1 Compact Spring Hanger

In the case considered above, several critical conditions were detected in the design phase and led to the choice of the CupDisk device instead of traditional spring supports. A key element in the choice was the compact dimensions and the reduced height of the mechanical component, which is less than half the height of a standard support with a similar load. This feature reduces the installation spaces and makes the system more stable in case of unexpected lateral movements. The technology was developed and patented by Saipem.

Fig. 13 - Parallel disposition

Fig. 14 - Spring-rate characterization

The devices were positioned in a section of the vertical piping portion and in correspondence of the Y-Tee, as indicated in figure 13. Detailed drawings of the supports and of the accessories are also provided.

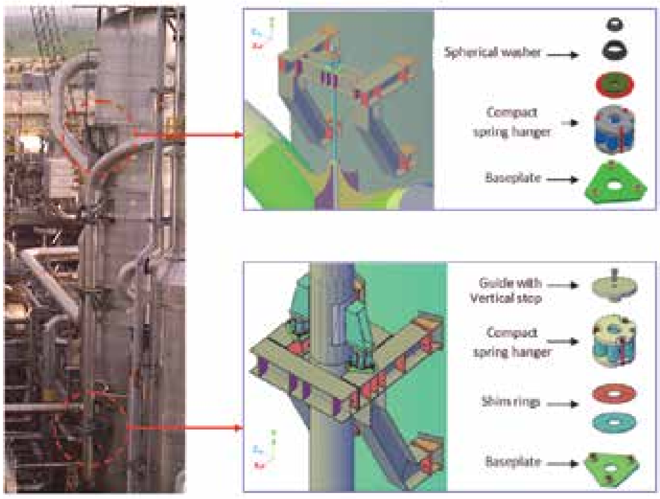


Fig. 15 - Constraints with compact spring hangers

The compact spring hanger has several features that make it different from other commercial products:

- The device is a compact variable spring support that operates in a range from 20 kg to 20 tons with the same dimensions.
- It has a modular design that allows installation in parallel or in series modules to increase either the travel or the load capacity of the device;
- Its compact envelope, which is independent from the travel and/or load setting, makes it suitable for installation in narrow spaces and especially under massive equipment;
- It can be equipped with many accessories or ancillary equipment to suit needs that may arise during installation (sliding pads, insulation slabs, tie rods, etc.);
- It is provided with an automatic selection software allowing the user to predefine the CD characteristics and installation strategy (two or more CDs in series or parallel).

Fig. 16 - Site Installation

4.4.2 Elbow Supports

A critical point of the circuit was represented by the elbows positioned at the bottom of the piping circuit, where stress concentration shall be avoided. Special clamped supports were designed in order to ensure a uniform distribution of the loads and to provide structural protection to the elbows avoiding any welding on the elbow.

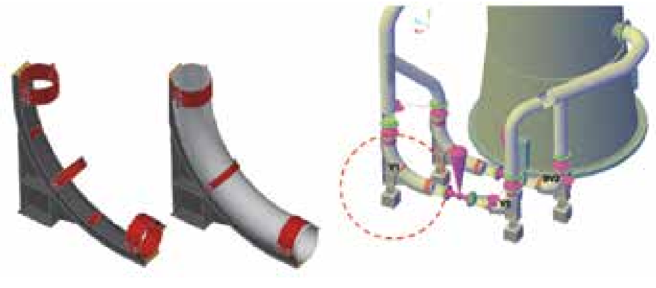


Fig. 17 - 3D-Model of elbow supports

The absence of welds between the elbow and the support prevents potential cracks generated from vibrations of the piping system.

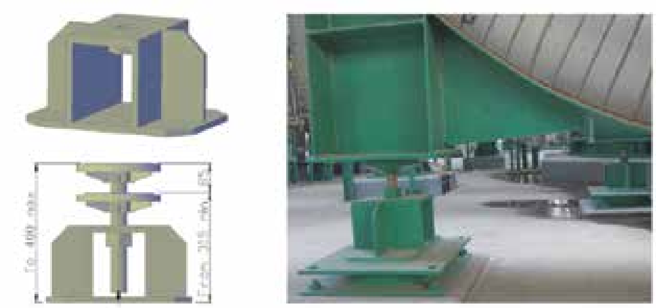


Fig. 18 - Vertical adjustment system

Vertical adjustment systems with PTFE sliding plates were added at the bottom of the support, providing a bearing plane and enabling horizontal sliding.

4.5 Nozzles

Once the mechanical analysis of the piping system was performed, the loads transferred to the vessel nozzles for each given load case were evaluated. The structural verification of the nozzles was developed by the Manufacturer of the vessel, with a FEM model applying the loads in accordance with ASME VIII Div.2 App.4 criteria and requirements. For material and allowable stress values the considered code was ASME SEC. II, PART D.

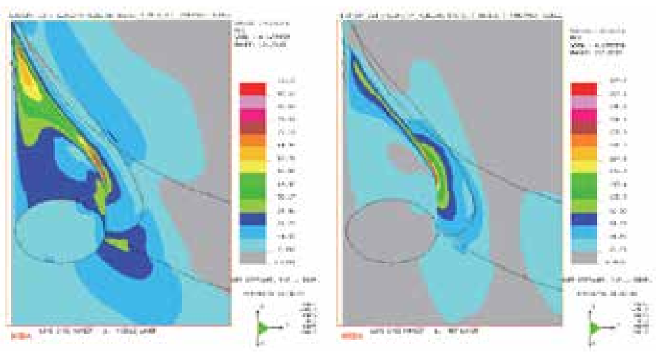


Fig. 19 - Nozzle stress (by Manufacturer)

4.6 Valves Group

Vibration phenomena at the bottom of the circuit should be avoided through the correct setting of the control valves groups in correspondence of the by-pass system. The solution consisted appropriately designing the control valves to prevent the occurrence of Flow Induced Vibration phenomena. The main parameters that were considered were:

- Vapour Pressure
- Pressure Drop
- Valve Opening

A key role is played by the actuator model, which was selected in order to control the fluid trim velocity, thus avoiding vibration problems. As for ISA recommendations, trim fluid velocity is 30 m/s for mono-phase fluid and 23 m/s for bi-phase fluid. In this example the actuator model selected is the one illustrated in fig. 21, with design maximum flow rate and the assigned valve opening.

Fig. 20 - Actuator model

If the maximum design flow rate is exceeded, the valve operates with an off-design opening and the pressure drop increases. The higher the pressure drops across the valve trim are, the greater is the risk of vibration. In this case the outlet pressure will become lower than the vapour pressure producing cavitation and flashing energy of the fluid at a higher-than-expected level. In such a condition, due to the increase of the volumetric flow and the bi-phase fluid, the valve CV will not be adequate and the trim fluid velocity will become higher than the recommended values, causing possible problems such as vibration, erosion, noise and FIV phenomenon on the piping system.

4.7 Summary of critical Issues and Design Solutions

A summary of the main critical issues and of the design solutions described in detail in the previous paragraphs follows.

	CRITICAL ISSUE	DESIGN SOLUTION
1.	PROCESS AND MECHANICAL REQUIREMENTS	Correct fluid path without occurrence of vibration phenomena and appropriate circuit lay-out characterized by mechanical reliability is achieved through the use of proper solutions: - Flow Rectifiers - Ad hoc Y-Tee device
2.	STATIC LOADS	Special support types were designed in order to ensure structural integrity considering the particular boundary conditions of the circuit: - CupDisk - Elbow Supports
3.	VIBRATION AT VALVES GROUP	Vibrations phenomena at the beginning of the circuit shall be mitigated by the correct design and setting of the control valve group.

5. WORKSHOP MANUFACTURING

Considering the criticality of this work phase a specialized workshop team was selected by Saipem in order to ensure high standards of production for the Y-Tee mechanical device. Saipem’s instructions, from first concept to final design, were followed in detail and led to the manufacturing process whose main steps are described below. Y-Tee was obtained from a single metal sheet, as all the components have the same thickness, as indicated in figure 21. The basic material used was ASTM A430 WP 316/316L.

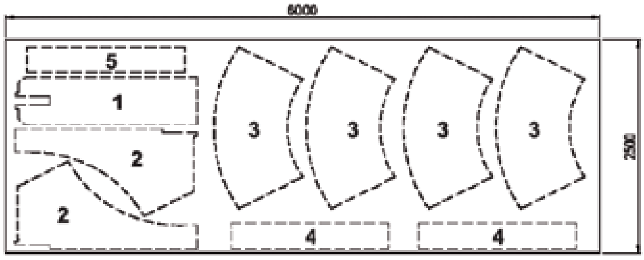


Fig. 21 - Components-Cutting from a single metal sheet

The fabrication started by cutting the plates with plasma technology and hot forming the shells for the elbows with a temperature range of 950-1150 °C. All the components (elbows, rings and crossbaffles and sub-components) were assembled by Shielded Metal Arc welding (SMAW).

Fig. 22 - Shell Hot forming

Fig. 23 - Lifting lug positioning

The root pass was performed using Gas Tungsten Arc Welding (GTAW). The main workshop phases are summarized below:

- Plate cutting
- Hot forming
- Rollering and weld-cut calking
- Calking of half shell: longitudinal section
- Calking of half shell: transversal section
- Welding preparation of cross baffles
- Welding of cross baffles
- Welding of half shell-baffles
- Welding of lifting lugs
- Welding preparation of connections
- Welding of connections and welding check
- Bench preparation for thermal treatment
- Thermal treatment in oven
- Sandblasting

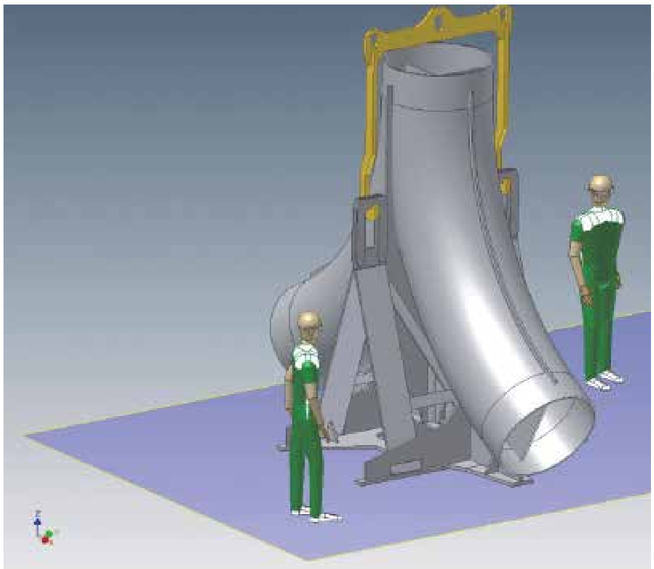


Fig. 24 - Temporary structure for thermal treatment

To perform the required final heat treatment with the annealing solution it was necessary to design a special temporary structure in AISI 304L.
The structure design took into account the decreasing material mechanical properties due to the solution reaching the high temperature at which annealing occurs (1050°C).
The final dimensional sizing was carried out using FEM analysis, considering the loads of the Y-Tee to be supported during the handling operations and the heat treatment.

Fig. 25 - Thermal treatment

Fig. 26 - Y-Tee dimensional inspection

Fig. 27 - Y-Tee transportation

After the heat treatment, all the welds were inspected by radiographic testing (RT) and by penetrant testing (PT). Finally the Y-Tee was subjected to dimensional inspection.

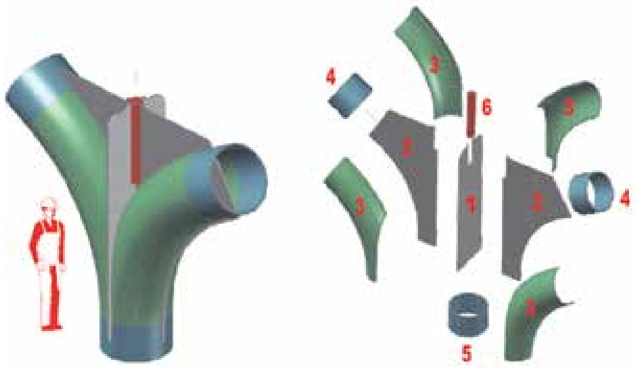


Fig. 28 - Y-Tee 3D model (assembled and exploded)

A further innovative feature of this component is its Modular design. The geometric parameters that may vary are: Diameter (DN), outlet junction angles (α1 and α2) and Thickness (t).

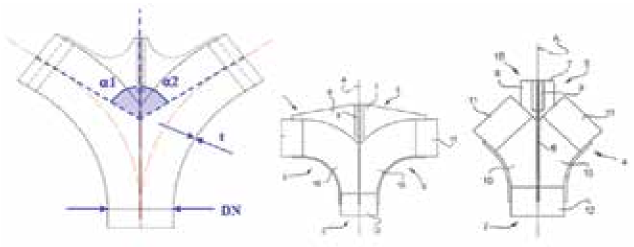


Fig. 29 - Y-Tee parameters (examples with different angle α)

This feature makes it possible to obtain the desired shape for every type of circuit path, without having to substantially vary the workshop phase.

6. ONSITE INSTALLATION

A significant aspect of the site erection concerned the installation of the Y-Tee piece, as it played a key role for the positioning of the whole circuit.

Fig. 30 - Y-Tee erection (step 1)

Fig. 31 -1st Branch erection (step 4)

Fig.32 - 2nd Branch erection (step 5)

Fig. 33 - Final Erection

A remarkable feature that distinguishes the mechanical component from other Y connections is the fact that it is self-bearing. The Y-Tee can be lifted, handled and kept in position during the erection phase without the use of other temporary support structures for the piping thanks to the lifting lug connection and the presence of stiffening cross baffles.
In addition the Y-Tee component was a template during the erection and connection of the: vertical inlet main pipe and the two lateral outlet branches. The device was directly suspended by a tie-rod connecting the lifting lug with the bracket.

6.1 Erection Sequence

In the following figure the main erection steps are represented:

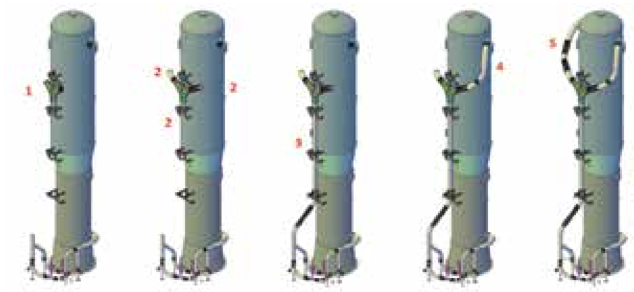


Fig. 34 - Erection sequence

STEP 1: Installation of the Y-Tee suspended with a provisional lifting-lug
STEP 2: Connection of the 3 spools by welding at the Y-tee
STEP 3: Erection of the vertical inlet main pipe, connected to the spool of the Y-Tee piece by means of a welded horizontal common plate, which blinds and subdivides the inner part of the circuit in two parts for the hydraulic test.
STEP 4: Erection branch 1 connection. First the part was connected to the nozzle (A1); because of the difficult 3D lay-out the adjustable spool has been used in order to obtain the exact design geometry, in accordance with the site condition. The on-site adjustment of the exact branch path has taken more than 2 weeks of working days. The shapes of the two lateral branches were machine-adjusted with the measured on-site geometry.
STEP 5: Erection branch 2 connection with the same procedure followed for the branch 1.

Fig. 35 - Blind intermediate plate

6.2 Hydraulic Test

The circuit was tested by subdividing the piping system into two parts separated from each other by a common welded plate, positioned in correspondence of the vertical guide C1.

Test 1

The piping is laterally guided on the positions C1, C2 and C, and is constrained in the vertical direction with a couple of temporary spools.

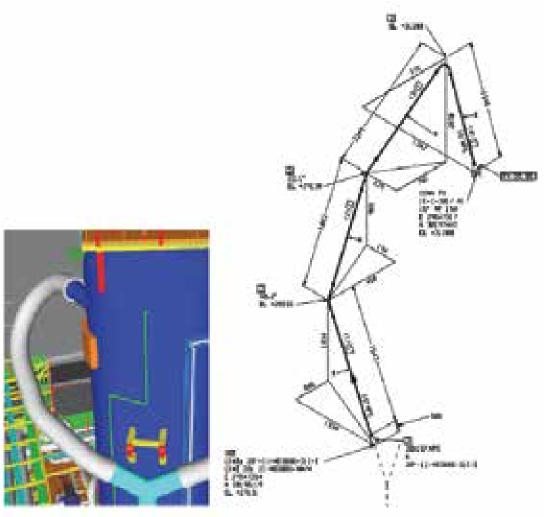


Fig. 36 - CAD-3D model and isometric sketch

The opposite end of the line has been blinded and connected to the nozzle of the HP flash Drum. The hydraulic test was performed at a pressure of 1.5 times the value of the design. After the test, the water has been discharged with a drainage hole on the bottom.

Test 2

For the vertical load, the Y-Tee has been properly constrained with a rope connected to the welded bracket structure on the vessel (position Y). The two branches have been blinded and connected to the tangential nozzles of the LP Flash Drum (point A1 and A2) and supported through proper scaffolding. The hydraulic test was performed at a pressure of 1.5 times the value of the design.

Fig. 37 - Structural temporary spools

6.3 Final Erection Activities

After the hydraulic test, the blind flanges applied to the nozzles A1, A2 and T2 were removed and the piping system was connected with the flanged joints to the Drums. As a second step, the blind common plate of the two sub-systems was detached and the two piping ends were joined by a circumferential welding (controlled with Penetrant Testing evaluation). The entire piping system was insulated with a coating and the temporary supports were replaced with the CupDisk at position Y and C and the locking pins were removed.

Fig. 39 - CO2 Stripping Column Effluent System

7. CONCLUSIONS

Saipem achieved the first commercial demonstration of a new invention, a significant technological challenge, with a focused management fitted to the entire process of designing, manufacturing and erection. The typical criticalities related to the effluent systems of ammonia flash drums were detected and a systematic approach that considered their interactions was followed. An analysis integrating both the fluiddynamic and mechanical aspects was performed aiming to define the most fitting solutions. The special Y-Tees and flow rectifiers, innovative technological devices, were successfully implemented. These solutions were the result of design choices defined ad hoc in order to ensure the proper process performance, preventing unbalanced fluid-dynamic induced forces, and at the same time guarantee structural integrity. The work was followed by a skilled team that combines the experience of Saipem process and mechanical specialists from various fields together with the staff on site in order to ensure that all the aspects of the project were completed with high quality standards. Also the manufacturing phase had a key role in ensuring that the devices possessed the required mechanical strength. A qualified workshop was thus selected so that the guidelines imposed by the company were correctly followed, with satisfying final outcomes. Building on Saipem's know-how and creating synergies among all the specialists led to the integration of their skills and expertise: the result is a project with innovative technological solutions that provides a reliable circuit for the plant.

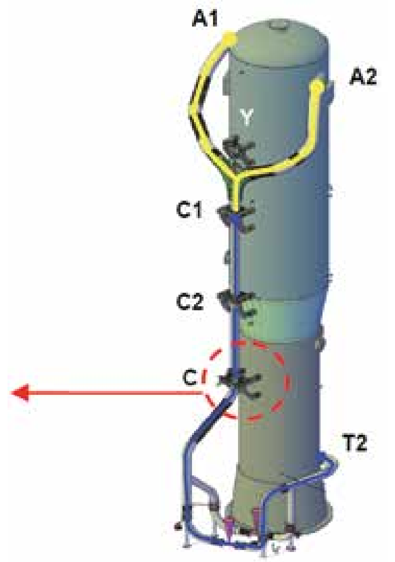


Fig. 38 - Circuit Subdivision for hydraulic test

A work-flow process, with the main design factors to be taken into account, was created and can be adopted by the specialists. It was therefore possible to capitalize on the experience gained from the various stages of the project, which will allow the application of the standardized guideline with high benefit in future applications.

The technology described has been successfully applied by Saipem in the two latest and world largest Ammonia-Urea single train plants that have been in operation for several years, proving numerous advantages over earlier designs and today this technology is available for incoming projects.

Paolo Minola, Saipem Spa

Petrovalves: a continuous effort into development

PetrolValves was born in 1956, at the peak of European post war industrial development. Since the very beginning it was clear how important is to have the most advanced production systems, in order to fully satisfy the customers' requests in term of quality.
This is when the valves production begins: from those of reduced dimensions to those with 110" diameter, still in production. In1960s PetrolValves was already recognized as one of the main European company producing valves for oil and gas industry; this position in the market has been reached also thanks to the patent of a new type of valve, later become standard for the market. Gradually the development of new products has marked the activities of PetrolValves, which has become a benchmark for valve industry. Thanks to its widespread presence in the five continents and an exceptionally large range of production, PetrolValves proves to be one of the main and most qualified valves producers all over the world.

PetroValves distinguishes itself mainly for supplying valves designed for special applications, where high engineering and performing standards are requested , with a large use of sophisticated technologies.
The company's manufacturing system includes valves for: extraction and transportation of hydrocarbons (subsea, off-shore, on-shore, pipelines), treatment and refinement systems, cryogenia, petrochemical industry, systems for energy generation.

"The use of a simulation code for supporting and valuing the Petrolvalves experience in supplying a high quality product is a solid practice of this company, build during the years thanks to big investments both financial and in human resources, then the decision to modify the consolidated methods for a new technology has been deeply meditated, taken satisfying some important requirements for both the code and the supplier: flexibility of solutions and resources, multidisciplinary of code and competencies, economic and company stability.
We chose ANSYS because it represents a reference technology in the world of FEM codes, whose integrated platform allows us to verify and implement our technical solution from a structural and a fluid-dynamic point of view; EnginSoft because with its twenty-years' experience in so different and complete applications it guarantees us an excellent quality of support and of flexible solution".

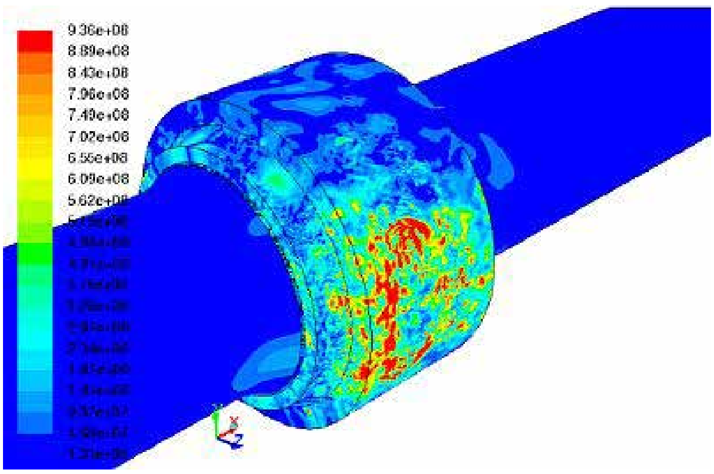
Ing. Pozzati
Technical Manager – PetrolValves

The constant investment in development of new technologies has helped PetrolValves to keep in step with the more and more pressing requests for performances and quality, sometimes even forestalling them.
Today PetrolValves produces valves which are the most technologically advanced, built with the most sophisticated materials.
The production has been continuously enlarged to satisfy the market's new demands of valves resistant to extreme temperatures (both high and low), very high pressures, extremely aggressive fluids.

Design the valves in ANSYS

Checking the structural resistances and the components' endurance is a consolidated practice in PetrolValves, and the employment of ANSYS with its innovative methodologies for approaching modeling and simulation problems, has permitted to reduce times of models' design, granting at the same time a more faithful representation of physical phenomena which rule the thermal and mechanical behavior.

Complete systems are analyzed, considering advanced aspects as the contact between the components, the loads in tie-rods, the couplings with grip, the effects of local plasticity of materials.
Moreover the post-processing of results can be customized on the base of the requirements and consistent with the current regulations.
The new technologies implemented in ANSYS have allowed to considerably increment the reliability of calculation with an excellent correspondence between experimental proofs and simulated data.



GE Oil&Gas: imagination at work

GE Oil & Gas

For its distinguishing performance and creative spirit, GE has always been a symbol of progress. Founded in 1878, as Edison Electric Light Company by Thomas A. Edison, it created the system to supply light and electricity around the world. GE has today diversified its activities, operating in technology-based sectors (media and financial services) and it constantly aims at the creation of

products, able to improve daily life. From aircraft engines to power generation, till to financial services, diagnostics, TV broadcasts and plastics, GE is present in more than 100 countries in the world, with about 300,000 employees. GE Infrastructure is a global leader in the supply of technology-based products, intended for emerging and developing countries, such as aircraft

engines, power, “oil and gas”, railways and water treatment services and technologies.

Oil and Gas is part of the GE Infrastructure, providing a wide range of solutions for the management and production of traditional and renewable power, oil and gas-turbine power plants, and is able to meet the international demand for reliable and effective power.

Born in 1994, after the takeover of Nuovo Pignone (ENI), Oil&Gas, leading company for materials

“A daily use of a thermal and structural solver like ANSYS, advanced and reliable both in the linear and non-linear field and well integrated with the CAD environment, is a must for a company in the forefront of precision engineering like ours. ANSYS products have been used in our product design for over 20 years. We decided to support our several and historical licenses of ANSYS Mechanical, used through a Linux cluster, with ANSYS Parallel Performance, with some ANSYS ICEM licensees, for an high quality mesh of preset modules for CFD computation and for demanding models in the structural field, plus some ANSYS CFX licensees for various analyses in the thermo-fluid-dynamic field. EnginSoft has always provided us with a qualified support in the CAE sector, like hot-line support for the internal use of ANSYS products and helping us in effectively getting through our regular work-peak problems, by using qualified and efficient services of thermo-structural engineering and fluid-dynamics, covering also those sectors which are of great importance to us such as room acoustics.

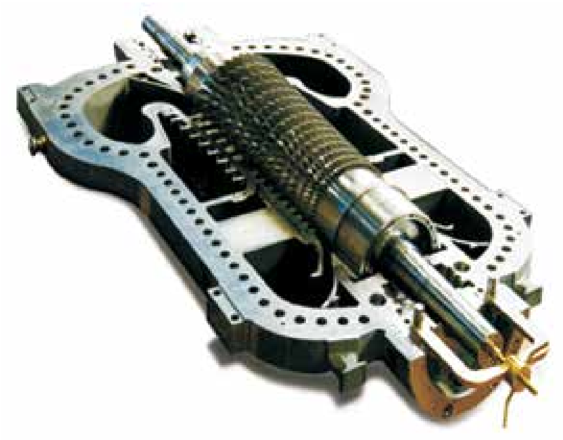
Mr. Stefano Generosi
Technical Computation Manager, GE Oil&Gas

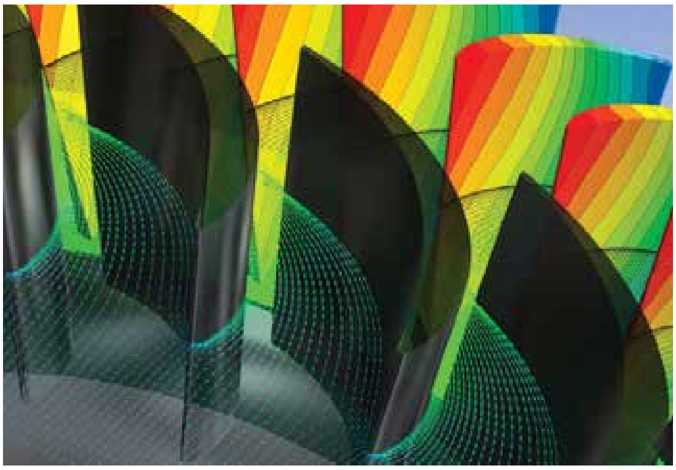
and services related to the turbine sector, is headquartered in Florence and represents GE main business in Italy. Oil&Gas offers, thanks to its research centres, devoted to the production and distribution of new products in Europe and United States, centralized productive solutions, LPG, transportation, storage, refinery, distribution and petrochemical systems, as well as, oil and gas pipeline integrity management. Technology and innovation are GE main objectives, aiming at leading new generation technologies. GE regularly works to anticipate and meet its customers' future needs and to turn creative ideas into innovative products and services, contributing to the resolution of some complex problems we now have to deal with.

Using ANSYS Mechanical, ANSYS CFX and ANSYS ICEM in design

ANSYS has always been an integral part of the mechanical design process of Nuovo Pignone, and when some years ago it was acquired by General Electric, such usual procedure for ANSYS products became even more common, being GE Group the bigger ANSYS customer in the world. Within GE, ANSYS products are used in the most different application sectors, and in GE Oil&Gas Firenze, several ANSYS Mechanical licenses are daily used to face complex thermo-structural problems related to the set of the produced turbines, as well as to analyze pressure containers, designed and produced in its plants located in Massa. The different applications range from simple static structural analyses in linear field – where its user-friendly approach and the parametric bi-directional interaction with CAD, allowed by the new ANSYS Workbench environment, are particularly appreciated - to complex non-linear thermo-structural analyses, dealing with creep phenomena comprising highly demanding geometries (turbine blades with several cooling channels) - where the power of the ANSYS mesher, with special regard to ANSYS ICEM can be highlighted, as well as to perform several thermo-structural analyses required by ASME rules for pressure containers and chemical reactors, where ad hoc features, always included within ANSYS, proved to be very useful to quickly work out any "linearization". ANSYS Inc. has recently implemented advanced capacity solutions in CFD field, by acquiring the leading product in this context, that is CFX, and integrating the new solver within the Workbench environment. Within Nuovo

Pignone, a new ANSYS CFX fluid-dynamic module, including the ANSYS ICEM advanced mesher has been satisfactorily used to face several problems that characterize the turbine sector in the thermo-fluid-dynamic field.






TOGETHER
along the innovation
pathway
Alongside our customers
we create new products
to breakthrough
future challenges
in the global
market
ANSYS®
ENGIN SOFT
www.ansys-italia.com | www.enginsoft.it
www.cobalto.it