Mechanical Engineering and CFD analysis Department

CFD and FEM Consultancy jobs descriptions

January 2020 Ed.
1. INTRODUCTION TO THE ENGINEERING DEPARTMENTS ................................................. 3
2. CONSULTANCY JOBS DESCRIPTION & PROJECT’S REFERENCE .......................... 4
   CFD STUDY OF A STORAGE AND PUMPING POOL .......................................................... 12
   CFD - DESIGN OF A DEFLECTOR WALL IN A PUMPS HOUSE ...................................... 13
   CFD VIBRATION ANALYSIS OF A HOWELL-BUNGER VALVE INSTALLED AT A DAM ............. 14
   CFD STUDY OF THE FLOW-RATE MEASUREMENT IN THE CANAL OF MELONARES DAM (SEVILLA) ...... 15
   FEM STUDY OF THE DAM DO BAIXO SABOR (PORTUGAL) ........................................... 16
   CFD STUDY OF CHANNELING INFRASTRUCTURE ............................................................... 17
   CFD STUDY OF THE ADVANCE OF SEDIMENT PARTICLES DURING WASHING PROCESS IN A STORM TANK ........ 18
   CFD SIMULATION OF TWO FIXED EFFLUENT AND ANALYSIS OF THE DILUTION OF ITS SALT CONTENT .......... 19
   CFD SIMULATION A SPILL FROM A SUBMARINE OUTFALL ................................................ 20
   FEM ANALYSIS OF AN OIL/SALT HEAT EXCHANGER ...................................................... 21
   FLOW AND AERATION CFD STUDY IN A WWTP REACTOR ............................................... 23
   FEM STRUCTURAL ANALYSIS OF A TRUNNION VALVE IN VARIOUS OPERATION CASES .......... 24
   CFD STUDY OF THE CV COEFFICIENT FOR DIFFERENT BUTTERFLY VALVES ...................... 25
   FATIGUE ANALYSIS OF A REACTOR USING THE NORM UNE-EN 13445-3 ................................. 26
   FEM STRUCTURAL ANALYSIS OF A FLOATING CAISSON (Break Water) .......................... 28
   CFD FIRE SIMULATION IN M-30 TUNNELS (MADRID) ................................................... 29
   CFD SIMULATION OF THE SMOKE TRANSMISSION IN A SUBWAY STATION ....................... 30
   CFD - WIND ACTION IN A WIND TURBINE FIELD .......................................................... 31
   FOUNDATION REPAIR FEM CHECKING OF WIND TURBINES .......................................... 32
   FEM ANALYSIS AND DESIGN OF A TURBINE FOUNDATION .......................................... 33
   DYNAMIC ANALYSIS OF WIND EFFECT ON A SOLAR TRACKER ...................................... 34
   CFD - WIND ACTION ON A SOLAR PLANT ..................................................................... 37
   CFD ANALYSIS OF WIND EFFECT ON AN HELIOSTAT ..................................................... 38
   CFD - FLOOD FLOW IN EL HIERRO RAVINE .................................................................. 39
   CFD - WAVE INFILTRATION STUDY THROUGH A SEA DIKE ............................................ 40
   FEM VIBRATIONAL ANALYSIS CRYOPLANT BUILDING ITER. FUSION FOR ENERGY ............ 41
   CFD ANALYSIS OF THE EVOLUTION OF AN EXPLOSION’S EXPANSIVE WAVE .................. 42
3. ADDITIONAL INFORMATION - SOCIAL NETWORKS ...................................................... 43
1. INTRODUCTION TO THE ENGINEERING DEPARTMENTS

Ingeciber has two engineering departments, the *Mechanical and CFD Engineering department* and the *Civil Engineering department*. The engineering departments have dealt with different projects using FEM and CFD software, such as structural analyses, heat transfer, fluid analysis using CFD software, electromagnetism, rigid/flexible solid mechanics – mechanical systems, etc. Some examples of the work performed by the *Civil Engineering department* are shown in following sections of this paper.

The FEM and CFD software available for the engineering departments to use to perform these analyses are: ANSYS, CivlFEM for ANSYS, CivilFEM powered by Marc, CFD++, FlowNex. ParticleWorks, SpaceClaim, Discovey Live and Discovey AIM.
# 2. CONSULTANCY JOBS DESCRIPTION & PROJECT’S REFERENCE

## Mechanical Engineering Department

- Supports & Frameworks
- Outstanding Structures
- Machinery’s specific components
- Linear & non-linear buckling
- Fatigue analysis
- Blast furnace components analysis
- Industrial plant components analysis following regulatory compliance
- Maximum stresses analysis following ASME VIII Div. 2
- Fatigue analysis following ASME VIII Div. 2
- Análisis de Uniones Faldón Envolvente.
- Thermal exchanger, aircooler and condenser analysis
- Pressure Vessels
- Analysis following regulatory compliance (RCC-MR, EN-13445, etc.)

## Non linear analysis

- Non linear materials (plascity, hiperelasticity, etc.)
- Structural instabilities (non linear buckling)
- Friction and frictionless contacts between components
- Geometric non linear analysis
- Composites
- Bolt analysis

## Dynamic analysis

- Modal analysis
- Armonic analysis
- Lineal and non linear transient analysis
- Spectrum analysis
- Free vibration analysys (PSD)
- Rotordynamic

## Thermal and CFD analysis

- Thermal isolation analysis of Pipes
- Thermal isolation design of train black boxes: Outstanding solutions.
- Thermal cycle simulations in air cooling systems for automotive industry.
- Testing storage design for dissipation system and material properties characterization.
- Thermal analysis of valves.
- Fire fighting door thermal-structural analysis.
- Electric refrigeration (CFD analysis)
- Thermal analysis of cryogenic systems

## Electromagnetic analysis

- Reactances.
- Permanent magnets.
- Coils, Induction engines
- Conductors, Wires.
Support analysis
- Highest loads supported
- Thickness estimate
- Design validation
- Cross section validation

Framework analysis
- Highest loads supported
- Thickness estimate
- Design validation
- Verification with UNE-EN 12663
- Tensile-compresive calculation

Outstanding structures
- Design validation
- Verification with building norms
- Last state analysis
- Opening process
- Checked with wind-tunnel results
Non-linear analysis

- Plasticity
- Hiperelasticity
- Composites
- Plastic deformations analysis
- Limit load analysis
- Hertz contacts

Linear & non-linear buckling

- Structural instability analysis
- Limit load in lineal buckling analysis
- Buckling Eigenvalues & Eigenvectors
- Non-linear buckling load estimate

Pretension bolts

- Bolt dimensioning
- Pretension loads analysis
- Sequential pretension analysis (various bolts)
- Gasket analysis
- Bridle verification
Dynamic analysis

Modal analysis
- Eigenvectors & Eigenvalues
- Modal deformed shape
- Participation factors
- Mass involved with each mode
- Prestressed modal analysis

Seismic design criteria
- Spectral analysis
- Seismic design criteria of Extraction Bombs, Furnaces, Pressure Vessels, Nuclear vessels, Structures, etc.
Maximun stresses analysis following ASME VIII Div. 2

- Maximun membrane and membrane+bending stress analysis following ASME
- Load combinations following ASME
- Thickness verification and reinforcement design following ASME

Fatigue analysis following ASME VIII Div. 2

- Fatigue verification following recomended ASME VIII procedure
- Load combinations following ASME
- Thickness verification and reinforcement design following ASME
- Cumulative fatigue usage analysis

Vessel to skirt joint analysis

- Maximun stress verification following ASME VIII
- Fatigue analysis following ASME VIII recomended procedure
- Load combinations following ASME
- Thickness verification and reinforce design
Thermal exchangers

- Maximum stress verification following ASME VIII
- Fatigue analysis following ASME VIII recommended procedure
- Load combinations following ASME
- Thermal analysis
- Thickness verification
- Tubular shells validation

Analysis following specific norms

- Maximum stresses verification following specific norms: RCC-MR, EN-13445, etc.
- Load combinations
- Thermal analysis
- Thickness verification
- Reinforce design
- Hydraulic pressure tests
- Outstanding structures
• Thermal isolation optimization. Low & high temperatures. Material properties temperature dependent.
• Analysis of thermal stresses
• Thermal isolation of furnaces and pipes
• Thermal isolation of tubes and anchoring for cryogenic systems
• Thermal analysis of cryogenic processors
• Thermal design of train's black boxes (phase change).
• Thermal cycle simulations at industrial components. Thermal stresses.
• Radiator analysis. Air-cooling for automotive industry.
• Testing storage design for disipation system and material properties characterization.
• Losses analysis
• Thermal disipation analysis in electric systems.
• Fluid-structure interaction.
• Industrial thermal exchanger analysis. CFD analysis.
• Fluid mixing analysis.
• Pipe pressure loss.
Electromagnetic analysis

Reactances
- Harmonic currents
- Eddy currents
- Losses
- Obtaining B & H fields
- Non-linear analysis (B-H curve)
- Thermal analysis
  - Joule Heat
  - Lorentz forces

Permanent magnets
- Coercitivity
- 2D & 3D analysis
- Vector potential analysis
  - B & H fields
- Ferromagnetic materials

Cable & Conductors
- Harmonic currents
- Different materials (Steel, Al, etc)
  - Skin effect
  - Joule heat
- Thermal analysis
  - B & H fields
CFD STUDY OF A STORAGE AND PUMPING POOL

A deflector wall was placed at the entrance of a sea water storage and pumping pool. To homogenize the flux, two apertures were created in the sides of the inflow catchment area. The operation of the pool revealed a problem in the bottom covering just in front of the lateral apertures. The objective of the simulation was to determine the causes of the failure and to determine the optimum operation level to avoid the lateral flux damaging the covering.

A sequence of CFD simulations with different operation modes was proposed, varying only the water level on the pool. The water level determines the mass of water in the inflow catchment area. Its interaction with the incoming flux disturbs the proportion of water which is alleviated through the lateral apertures and the velocity of this flux.

This study, which was performed with ANSYS-CFX, allowed us to understand the origin of the problem detected and to define the optimum water level to guarantee the correct hydraulic behavior of the pool.
**CFD - DESIGN OF A DEFLECTOR WALL IN A PUMPS HOUSE**

The modification of the mode of operation in a house of pumps is projected. The adding of a pump modify the flow established in the large pitcher and turbulence arising are sucked by the pumps compromising its operation.

A CFD simulation is planned with a three-dimensional model showing the flow behavior of the water in the pump housing and the different options for a deflector wall to optimize the overall performance, considering the various modes of operation raises the large pitcher and boxes of bombs.

The results show how a small change in the length of the wall or on exit angle, alters the flow behavior across the large pitcher. After several iterations in the design process and analysis, the design of the baffle wall is optimized.

**Phases:**
- Generation of three-dimensional geometric model of the large pitcher and boxes of bombs.
- Domain fluid extraction study.
- Domain meshing with refinement of boundary layer.
- Preparation of multiple technical solutions with different deflector walls.
- Definition of the boundary conditions which simulate the different operation modes.
- Results comparison.
- Design optimization of the deflector wall.
- Validation of the solution adopted verifying its behavior in all modes of operation.
CFD VIBRATION ANALYSIS OF A HOWELL-BUNGER VALVE INSTALLED AT A DAM

The goal of this study is to analyze using CFD techniques the hydraulic working of the Howell-Bunger valve placed at the Aguilar de Campoo (Palencia, Spain) Dam, which is subjected to unusual vibrations.

Three phases:

- **Phase 1**: Study of the actual configuration of the pipe and the valve
- **Phase 2**: Study of the pipe with some correcting measures (guides)
- **Phase 3**: Optimization of the installation place of the guides

Turbulences on the pipe
CFD STUDY OF THE FLOW-RATE MEASUREMENT IN THE CANAL OF MELONARES DAM (SEVILLA)

The purpose of this study was to obtain the depth-flow curves in the intake canal of the Melonares dam. It is the first study of its kind conducted by a CFD model, without making a reduced model.

Simulation phases:

- 3D model of the canal, from the pumping station.
- Transient and multiphase simulation of pumping process to stabilize the flow in the system.
- Results analysis and obtaining of the flow-rate measure curve.
FEM STUDY OF THE DAM DO BAIXO SABOR (PORTUGAL)

The dam *do Baixo Sabor* is an arch dam placed on the north of Portugal. It is a tall structure in which is necessary to analyze the concrete temperature evolution during the construction considering different cooling and environmental hypothesis (cool water, river water or both).

The cooling system consists of the arrangement of coils. The purpose of this study was to dimension the cooling system.

A thermal transient and evolution calculation was performed taking as reference the values of the permissible thermal indicators proposed in the *Technic Guide nº 2* of the *Spanish Committee of large dams*.

*Thermal gradient in one of the phases of construction*
CFD STUDY OF CHANNELING INFRASTRUCTURE

The predicted works for the channeling of the water in Argamasilla ravine included different secondary streams. The aim of the channeling is to alleviate the constant floods that regularly affect the city of Écija (Sevilla).

The 2D simulation didn’t allow us to tackle the complete channeling analysis. Furthermore, it was necessary to analyze the transient evolution of the incoming flood and not assume a steady state flow. For this reason, a tridimensional study of the incoming flood was modeled with ANSYS-CFX, taking into account the elevated water level in the Genil River, which is where the simulated channeling flows into.

The simulation shows the behavior of the flood in a transient regime, with a return period of 50 years in the last section of the Argamasilla ravine channeling. The results showed that the initially projected solution did not take into account the behavior of the flood at the initial moment. When the flood finds its outlet blocked by the Genil River, the water impacts against the upper wall and emerges through the upper ventilation grids, creating a water jump which returns back through the internal part of the duct. This study and its results were used for the correct installation redesign.
CFD STUDY OF THE ADVANCE OF SEDIMENT PARTICLES DURING WASHING PROCESS IN A STORM TANK

The construction of an underground storage tank for storm water is projected. A high level of fouling due to sedimentation while water is stored is envisaged. To lower maintenance costs, a washing process is designed without the direct action of operators, consisting of a sudden emptying of a depot situated at the top of the tank, hoping the current to pull out and drag the sediments deposited.

Phases:

- Generation of geometric model.
- Transient study of flood flux.
- Discretized study of particles advance
- Quantifying the degree of final advance of the particles along the storm tank.

In the first phase is studied the flood process caused by the discharge of header tank. Transient flow simulation allows determining the hydrograph of the flood, which is shown superimposed on the image. Known the induced trawling by the flood, is discretized spatially and temporally the phenomenon of entrainment of particles and quantified progress in each state of the flow of the flood. The results showed that the sediments are dragged from the first half of the tank, but these are accumulating at the back and a few leave the tank.
Plants for desalination of sea water pouring require the extracted salt treated water. In this process, it is necessary to control the final discharge salinity lowering it by mixing it with seawater. The mixture of effluent with seawater in the same pipe is not enough to guarantee the complete homogenization of salinity. The objective is to simulate the mixing of the effluent with sea water, taking into account the geometry of the union and the discharge pipe.

The analysis focuses on quantifying the evolution of salinity at each point of the pipeline. An added difficulty is that the injection of the discharge on the side of the pipe do not mix the two effluent because no turbulence is formed, and the homogenization has to be produced by diffusion and stirring derived by the presence of bends in the path of the pipe.

**Simulation Phases:**

- 3D model of a 250 m discharge pipe considering the blend placed at 50 m from injection point.
- Exhaustive meshing of the fluid domain with hexahedral elements that allow the collection of all the details of the salt diffusion.
- Application of models of particle concentration and diffusivity models.
- Multiple simulations applying different salt concentrations and flow rates to represent the different operating modes.
- Analysis and quantification of mixed homogenization of salinity in the final discharge.
The objective of this study is to monitor the spill of sewage water from a submarine outfall to the ocean. This type of study is very important for complying with the environmental regulation.

A model of the shore with the submarine outfall is created. Due to the difficulty of anticipating the spill behavior a big domain was simulated. The adaptive refinement of the software XFlow allows controlling this type of domain without wasting computational resources. Scalar transport equations were used for the simulation and monitoring of the spill.

An environmental regulation was considered, checking the maximum fraction of sewage water at a distance from the outlet of the submarine outfall. With a spherical limit surface it was checked the position of the isosurface of maximum fraction allowed by the environmental regulation.
FEM ANALYSIS OF AN OIL/SALT HEAT EXCHANGER

This project is the analysis of hot oil to molten salt heat exchanger located in Morocco after several heat exchanger welds installed at the Solar Power Station have failed. The design of this heat exchanger, focusing in the failure area, is analyzed by ANSYS by the Finite Element Method (FEM) under the design and operation conditions defined by the client, studying the results with the ASME VIII Div2 code.

Three different cases have been studied:
- The equipment design successfully fulfills the ASME code stress linearization criteria.
- A transient simulation has been performed. A stress maximum has been measured at the critical weld for an intermediate time (due to the temperature drop between cavities).

- A fatigue case, considering salts loading and unloading. The maximum stress increment at the critical weld has been measured for an intermediate time (due to the temperature drop between cavities). Following the ASME code the equivalent stress has been obtained and introduced in the S-N curve. A total of $15.7 \cdot 10^6$ cycles has been obtained under fatigue at the critical weld, being higher than the minimum number of cycles defined by the client (22000 cycles).

It can be concluded that the critical zone of the heat exchanger has been designed correctly under the defined loads considering the stress and fatigue criteria of the ASME code.
FLOW AND AERATION CFD STUDY IN A WWTP REACTOR

The design of a reactor of a Waste Water Treatment Plant (WWTP) has to focus towards a uniform movement of sludge to prevent formation of prone to sedimentation and proper aeration dead zones to encourage aerobic digestion of organic wastes. A design raised in carousel reactor with an aeration system consisting of surface aerators, is expected that the surface agitation of the sludge allow enter enough air into the device.

Phases:

- Generation of three-dimensional geometric model of the WWTP reactor with flowing accelerators and surface shakers.
- Fluid domain extraction, establish boundary conditions and fluids provided for installation devices.

As a result the flow characteristics established and dragging the oxygen introduced by surface shakers shown. Aeration is very localized and insufficient, because below the surface there are shakers much of the flow is not aerated.

Following this study finding alternative was approved for a more effective aeration.
FEM STRUCTURAL ANALYSIS OF A TRUNNION VALVE IN VARIOUS OPERATION CASES

The purpose of this study was basically to verify structural parts of a trunnion valve under different operational loads such as internal pressure, bolt preload, opening and closing torque, etc.

Different structural loads in the main parts of the valve were analyzed. Nonlinear contacts were used to gather the separation effects caused by the pressure on the bolted flanges.

An interesting part of the study was the analysis of the closing ball subjected to the effects of the pressure that is compressing an internal hyperelastic gasket.
CFD STUDY OF THE CV COEFFICIENT FOR DIFFERENT BUTTERFLY VALVES

The purpose of this study was to obtain the Cv coefficients for a set of butterfly valves for different angles of the disc or “butterfly”. This study was done for 15 valves of different diameters and simulating 9 different opening angles.

Due to the high total number cases for being simulated (135 cases), the study with a traditional mesh based CFD software implied an important extra time for meshing each case, which implied a price increase.

The simulation was done with XFlow, based in the Lattice Boltzmann method, which avoids the mesh creation. That fact allowed the automatization of all the simulations with the parameters obtained for the first case, also using the advantage of easy geometry exchange between cases.

The chosen simulation tactic implied the obtaining of all the Cv coefficients for all the studied valves in a time much more reduced than the possible with a traditional mesh based CFD, with the consequent price save for the client.
FATIGUE ANALYSIS OF A REACTOR USING THE NORM UNE-EN 13445-3

Fatigue analysis performed by ANSYS of a reactor applying the norm UNE-EN13445-3 by using the Finite Element Method.

It includes a variation in the coil design near the supports and the changes implemented in a previous review: variation of the boundary conditions of the supports, allowing its radial displacement with the expansion of the equipment, the variation of the lifting lugs, the variation of the stiffness plates of the nozzle B3, the variation of the temperature and pressure loads in the coil and the introduction of a modeled weld in the reinforcement of B3.
Figure 5.2 Displacements detail. Case 1

Figure 8.6 Stress increment 2 in coil
FEM STRUCTURAL ANALYSIS OF A FLOATING CAISSON (BREAK WATER)

A 3D concrete floating voided caisson is modeled in order to optimize the thickness of the concrete elements and reinforcement amount. Different calculation models are studied for each stage: transport, positioning, immersion and service.

The loads taken into account and the combined for ultimate and serviceability limit state are the ones considering all stages including pressures due to immersed caisson (ballasted with water or with sand).
CFD FIRE SIMULATION IN M-30 TUNNELS (MADRID)

M-30 is composed of two parallel underground tunnels over 50 meters that withstand a huge transit of vehicles. Ventilation inside the structure and proper evacuation of the smoke from a possible fire proved vital.

The following calculations were made:

**Simulation phases:**

- 3D modelling of the ventilation system.
- Transient simulation with variation of pulse energies and extraction based on the evolution of the fire.
- Smoke spread simulation.
- Analysis of results and corrective actions.
CFD SIMULATION OF THE SMOKE TRANSMISSION IN A SUBWAY STATION

The aim of this transient CFD analysis is to study the transmission of the smoke inside a subway station due to the fire produced in a train. This type of study is very important inside the security studies which must be done during the design phase of the station.

Scalar transport equations were used for simulating the smoke, being its movement governed by the buoyancy effects due to the differences of temperature and density.

During the study the quantity of smoke was monitored in the exits and other points of the station. With these results it was checked which were the escape routes in which the smoke maximum value was produced before.

This study showed the need of applying extra security measures in some routes of the design to avoid the fast concentration of smoke.
CFD - WIND ACTION IN A WIND TURBINE FIELD

The aerodynamic behavior of a wind turbine field is conditioned by the adjacent terrain. The register of the wind in the zone is limited and the wind tunnel studies do not capture the orography characteristics with the presence of moving wind turbines.

To obtain more accurate and realistic results, tackling the study with computational fluid dynamics (CFD) techniques was proposed. The conventional software with their traditional approach, based on the finite volumes method, are not sufficient to accurately study such a big domain with the presence of wind turbines in movement.

The software used was XFlow, with the assistance and knowledge of the Mechanical Engineering and CFD Department.

The topographic surface of the area was introduced in the software and the 28 wind turbines which form the wind turbine field were placed accurately. The extension of the terrain studied was 432 km². The rotation degree of freedom of the blades was allowed and the wind was defined with a real wind profile of the area. The adaptive refinement of the lattice allowed the capture of the movement of the blades and their influence on the wind distribution.
FOUNDATION REPAIR FEM CHECKING OF WIND TURBINES

Foundation repair checking of wind turbines in Pedregoso wind park.

The study is focused in the foundation critic zones checking. These areas are more susceptible to damage. This study put the focus on the part of the original foundation where the active bars are anchored guarantying the union between the design repair and the existing foundation.

These results are compared with the repair-less foundations, with the goal of determining their ultimate load and the necessity of being repaired or improved.

An static and nonlinear FE model of the existent foundation has been modeled corresponding to a standard wind turbine. This model takes into account the repair proposed by the customer with the different load cases acting on the structure.
FEM ANALYSIS AND DESIGN OF A TURBINE FOUNDATION

Object
• Analysis and Design of Turbine Foundation

Calculations Steps
- Model used for dynamic analysis is taken for static analysis
- Material properties (concrete, steel), real constants, codes to be used (EUROCODES, ACI, etc.) are specified in CivilFEM
- Various load steps are specified in ANSYS and the load combinations done in CivilFEM
- Reinforcement design using CEB Code method
- Envelope of reinforcement design results over various load steps obtained (contour plots and listing is possible)
- Design optimization and shear check
DYNAMIC ANALYSIS OF WIND EFFECT ON A SOLAR TRACKER

This project was carried out to analyze the possible dynamic effects generated by the wind on a solar tracker structure, not contemplated in a conventional static analysis. This Report shows the results obtained in the analysis for the design wind situation once the structure has been modified for its fulfillment.

The Solar Tracker structure consists of a horizontal axis square profile tubing (torsion beam) supported by vertical elements embedded in the terrain. Panels are mounted upon the tube by a series of purlins to which the panels are attached. The axis of rotation is horizontal with respect to the ground and the motor is located at the center of the axis. The rotation is restricted at this point and it is free in the bearings on which the axis is attached to the posts.

CFD RESULTS
FINITE ELEMENT TRANSIENT ANALYSIS
After obtaining the first results of dynamic effects due to the interaction between the wind and the panel (angles and frequencies), the moments generated in the central tube were applied from these results.

From the transient analysis of the FEM model of the structure, the performance over time of the tracker's structure is calculated, obtaining the maximum tension and moment that they have during the first seconds.
CFD - WIND ACTION ON A SOLAR PLANT

Solar panels that make up a solar plant are exposed to weather actions, among which are the strong winds that occur occasionally. This was the case of the solar garden in Vejer de la Frontera, which in April 2011 suffered damage due to the wind.

The case of study is the wind around the solar array so that checkpoints where anemometers are placed get the same values than those observed in both speed and wind direction. In this situation, the wind speed on the exact location of the solar plant and solar panels affected is analyzed.

The results show that the surrounding orography alters the behavior of the wind and when the conditions observed in April 2011 are reproduce, the solar farm area is swept by winds of over 115 km/h, which exceeds the design strength panels.

Phases:

- Generation of three-dimensional geometric model of the terrain in the area of solar plant.
- Imposition of boundary conditions that define the observed wind and verification by sensors placed on the position of the reference anemometers.
CFD ANALYSIS OF WIND EFFECT ON AN HELIOSTAT

The design and dimensioning of equipment exposed to wind actions requires a thorough aerodynamic analysis. In the case of a heliostat, the large size of the mirror determines its wind resistance.

To address this analysis in time and reasonable price, the best option is the three-dimensional simulation using CFD programs. Two situations were simulated: the mirror in a vertical position subjected to a wind of 100 km/h, and the mirror in the safety position under a wind of 150 km/h.

Data of the drag, the tilting in the column base, and lift force and torque turning of the upper joint is obtained. These latest actions come from the deflection of the air flow when the mirror is folded.

Phases:

- Adequacy of generating three-dimensional geometry and fluid surrounding the heliostat domain.
- Meshing of the fluid domain with special caution in the details of geometry and elements of the boundary layer. The final calculation mesh consists of 22 million elements.
- Definition of wind action conditions.
- Resolution of the simulation and monitoring of the resultant forces.
- Analysis of results and conclusions
CFD - FLOOD FLOW IN EL HIERRO RAVINE

The project consist in the channeling the final stretch in the ravine of El Hierro. A previous study with a linear program predicts a proper operation of the infrastructure. But its design with sharp changes of levels and curves, imposed by the natural layout of the ravine, results that a linear software can’t take into account.

A CFD simulation is planned with a transient 3D model. Hydrograph of the design flood is introduced. The multiphase simulation reveals the real behavior of flood in the infrastructure.

The channeling gets pressure in several sections and water overflow in ventilation apertures.

With the obtained results it was checked that the construction the channeling does not meet the design criteria.

Phases:
- Generation of three-dimensional geometric model of the entire channeling and the aeration system.
- Removing the fluid domain and meshing thereof with special caution in the boundary layer meshing.
- Imposition of boundary conditions that define the input flow of design flood.
CFD - WAVE INFILTRATION STUDY THROUGH A SEA DIKE

The design and dimensioning of a sea dike to protect an artificial platform requires a stability study, overtopping waves and infiltration inside. To address this analysis in time and reasonable price, the best option is the simulation using CFD programs. Every detail of the different layers that make up the dike was introduced into the program, each characterized by its porosity and permeability. An area of open sea 300 meters long was generated and was coupled to the dike so that the water flows therein.

Simulations show the behavior of the waves in the sea, where the waves break by their interaction with the seabed, and the impact thereof on the dam and the subsequent infiltration. The results allow to analyze the behavior of infiltration and saturation evolution of the different layers and the groundwater.

Phases:

- 2D model of the different layers of the dike and 300m stretch of sea.
- Meshing of porous domains that form the dike and meticulous hexahedral meshing of the fluid domain that represents the extent of open sea with waves.
- Definition of the groundwater level and transient generation of waves.
- Resolution of the simulation and monitoring of the evolution of the water table.
- Obtained results analysis. Conclusions.
FEM VIBRATIONAL ANALYSIS CRYOPLANT BUILDING ITER. FUSION FOR ENERGY

The project consists in the vibration analysis in an ITER’s plant building produced by a set of rotative machines (compressors) for the final customer Fusion For Energy. The objective was to verify that the maximum amplitude of vibration in a range of frequencies was not over an allowable value for all the structural parts of the building.

As shown in the image at the left the stratum composed by rock (pink), soil (blue) and compacted fill (red) was modeled.

The amplitude of displacements, accelerations and velocities in critical measure points of the building where checked along the working frequency range of the rotative machines.
CFD ANALYSIS OF THE EVOLUTION OF AN EXPLOSION’S EXPANSIVE WAVE

The structural design of a building in an oil & gas plant has to take into account the dynamic behavior of the building due to an expansive wave produced by a near explosion.

The analysts have to measure the building to satisfy the codes. The critic point is to know the time evolution of the over pressure pulses and the places of the building more exposed.
3. ADDITIONAL INFORMATION - SOCIAL NETWORKS

More information about the different business lines of Ingeciber is available at the websites shown below:

- Ingeciber website: www.ingeciber.com
- Youtube Channel: Youtube channel
- CivilFEM website: www.civilfem.com
- Int’l UNED-Ingeciber FEA Master’s: www.uned.es/mastermef
- Int’l online CAE Education CENTER: www.icaee.com
- Ingeciber Linkedin: https://www.linkedin.com/company/ingeciber
- Ingeciber Twitter: @Ingeciber. https://twitter.com/Ingeciber
- Facebook: www.facebook.com/ingeciber/
- ANSYS Preferred partner: https://www.ansys.com/about-ansys/partner-ecosystem/software-partners/ingeciber