MULTIDISCIPLINARY SIMULATION OF AUTOMOTIVE WATER PUMPS

Remo De Donno
Industrie Saleri Italo S.p.A., Italy

KEYWORDS –
ANSA, EPILYSIS, META, automation, ANSYS, OpenFOAM, Dakota, Python

ABSTRACT –
Saleri has been using the BETA CAE products since several years to simulate automotive water pumps and cooling systems. CAE engineers use daily the ANSA pre-processor, which ensures a fast and reliable setup for CFD (OpenFOAM and ANSYS CFX) and structural (ANSYS Mechanical) simulations, including optimization studies (Dakota and EPILYSIS). Once the solution is ready, the META post-processor is able to automatically generate the results of interest. The purpose of this presentation is to show some key phases of the simulation workflows at Saleri, underlining the essential role of the BETA CAE products in achieving consistency, repeatability and robustness.

TECHNICAL PAPER –

1. THE SALERI GROUP

Saleri is a leading company in the design, development and production of water pumps and cooling systems for the automotive industry, whose logo is shown in Figure 1.
The main components of pump assemblies are shown in Figure 2: the pulley of the pump takes the rotational velocity from the pulley of the engine by means of belt connection. The motion is transferred to the impeller through the bearing; wet and dry parts of the machine are separated by means of a mechanical seal. The task of centrifugal pumps for the automotive field is to control the engine temperature by pumping the coolant inside the cooling circuit. The flow arrives at impeller from the engine suction pipe and, only when the coolant temperature is high, also from the radiator suction pipe. When the coolant temperature is low, a thermostat closes the way to the radiator, forcing the liquid to flow through the short circuit in order to reach the design temperature as fast as possible. After passing through the impeller, the flow decelerates inside the volute where part of the kinematic energy is used for increasing the static pressure to the value necessary for overtaking the circuit resistance and pumping the coolant. The pump assembly of this example is equipped with a thermocouple connector for the sake of temperature measurements.

Figure 1 – Logo of Industrie Saleri Italo S.p.A.

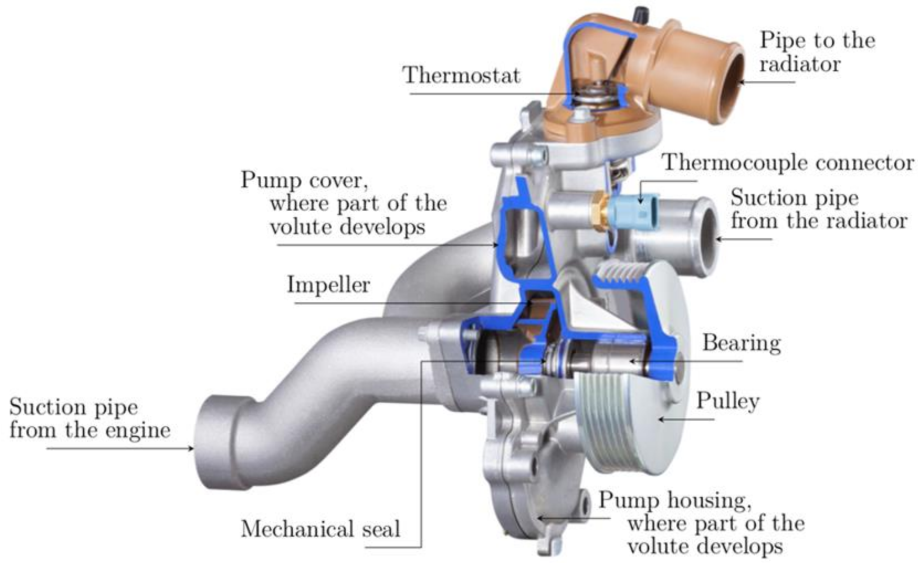

Figure 2 – Centrifugal pump assembly for the automotive field produced by Industrie Saleri Italo S.p.A.

<u>The role of simulations in Saleri</u>

Simulations at Saleri have a fundamental role in both design and verification phases. In fact, simulations are used since the very first phase of the project for designing the main components of the pump as well as at the end of the project for validating the machine, as shown in Figure 3. The simulations fidelity increases with the maturity of the project, from the concept phase to the series phase.

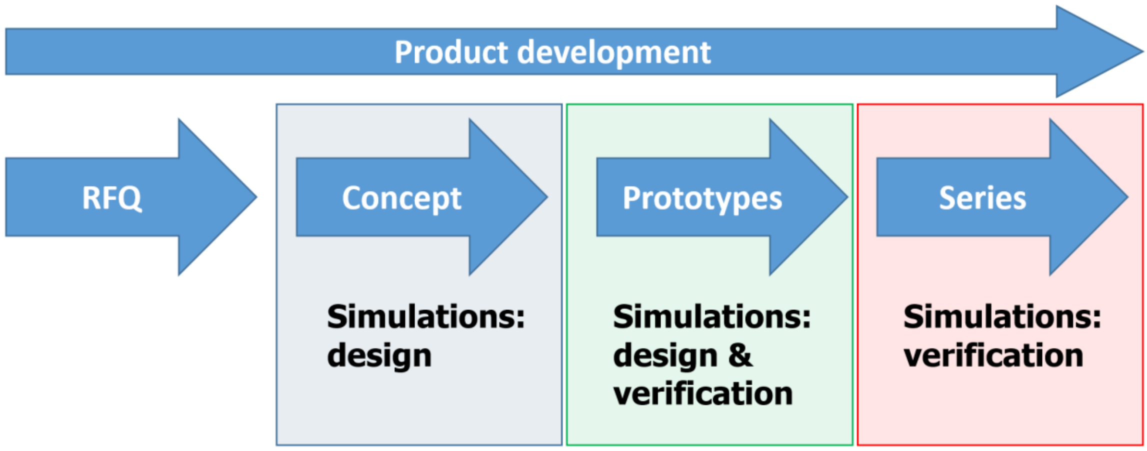

Figure 3 – The role of simulations in the project phases

2. CFD SIMULATIONS AT SALERI

CFD simulations consist of different phases. After cleaning and preparing the CAD geometry, i.e. the virtual test bench shown in Figure 4, the computational grid is generated. Figure 5 shows a shell mesh and a volume mesh of a real centrifugal pump, while Figure 6 shows the computational grid of a positive displacement pump. During the meshing phase, particular attention is dedicated to the near wall region, in order to obtain a proper y+ value.

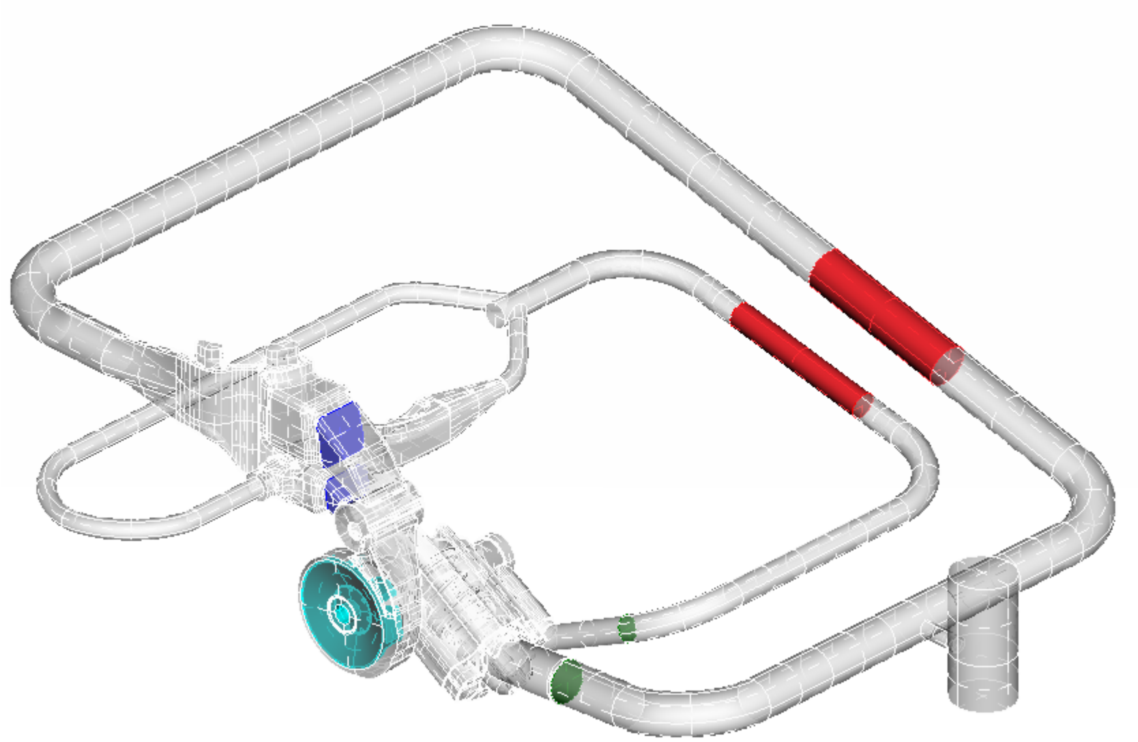

Figure 4 – Virtual test bench for calculating the pump performance, considering explicitly the engine resistances

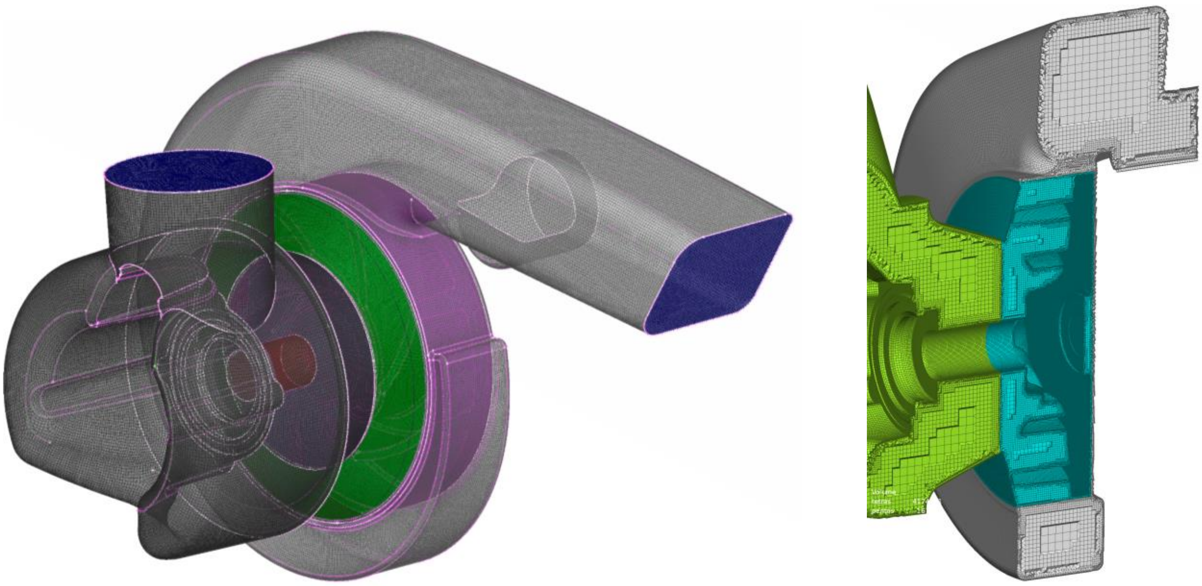

Figure 5 – Shell mesh of the fluid domain (left) and volume mesh of a fluid domain section (right) of a centrifugal pump

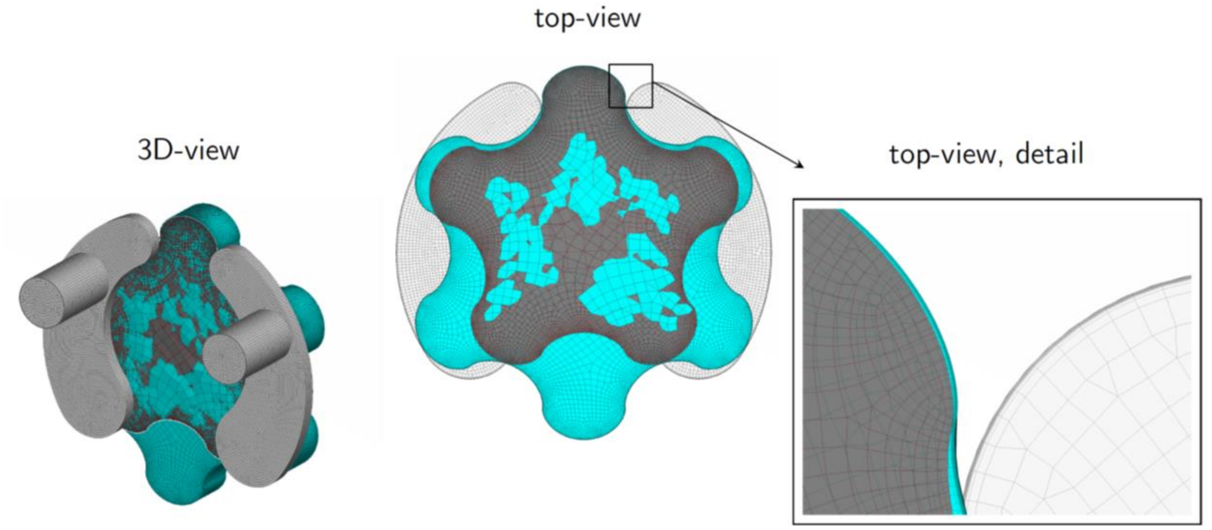

Figure 6 – Computational grid of a positive displacement pump (gerotor) for the immersed boundary method

After generating the computational grid, the solution phase is defined. Depending on the maturity of the project, different methods can be applied for the flow computation of centrifugal pumps. In particular, RANS steady-state simulations with frozen rotor or mixing plane approach as well as unsteady simulations with sliding mesh approach may be performed. The y+ is set in order to correctly predict the impeller torque and the SST-kω turbulence model is chosen for better consider potential flow separations.

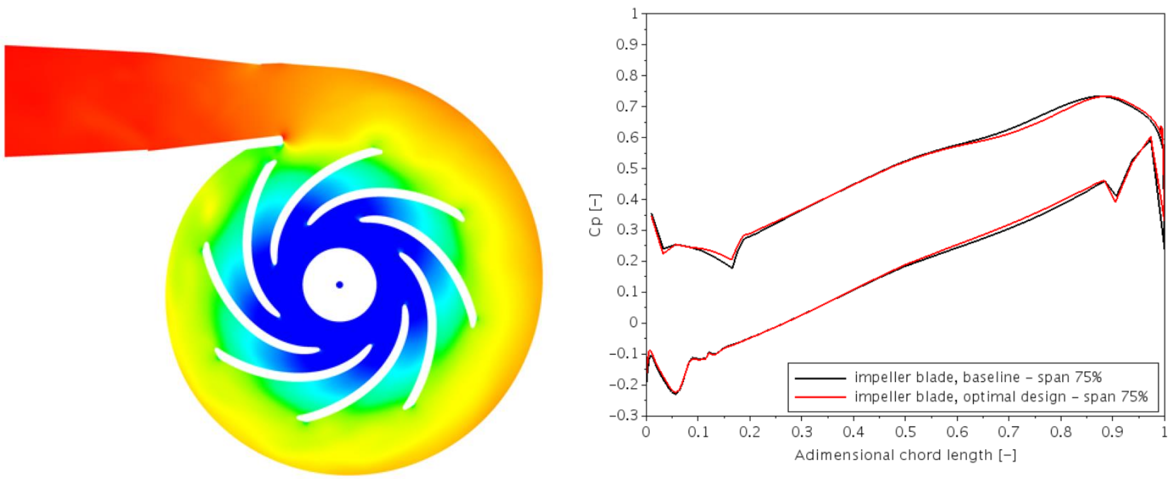

Figure 7 – Section view of a pump pressure field (left), blue indicates low pressure while red indicates high pressure. Pressure coefficient along the blade of a centrifugal pump at span 75% (right)

Once the solution is obtained, the post-processing phase allows to qualitatively as well as quantitatively evaluate the fields of interest. Figure 7 shows the pressure field of a pump and the pressure coefficients of a blade, while Figure 8 shows the streamlines of a pump and the vapour volume fraction at the impeller when cavitation occurs.

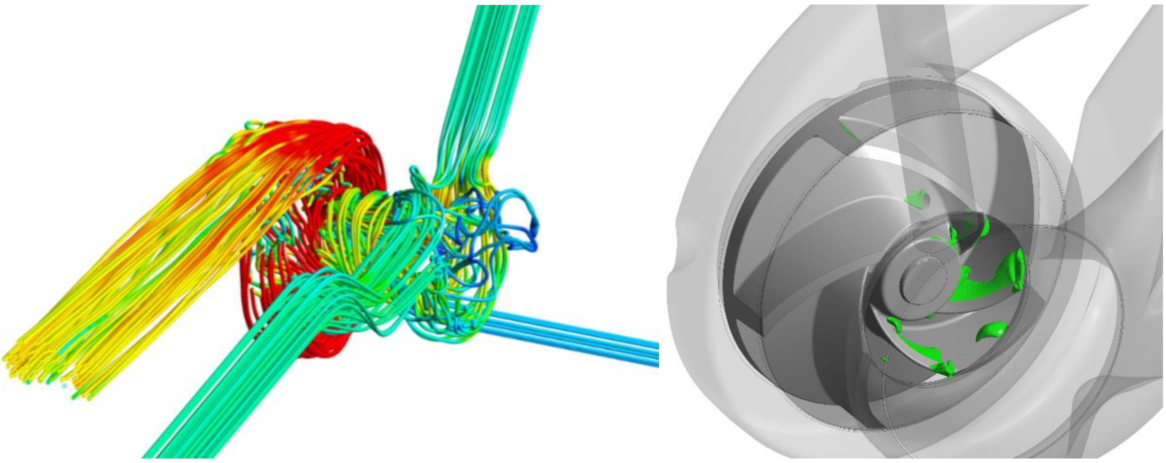

Figure 8 – Streamlines (left) and vapour volume fraction (right) of a centrifugal pump

<u>The role of BETA CAE software packages</u>

All the pre-processing phases of the CFD simulations described above are performed with ANSA. The CAD geometry is imported in the software, is cleaned with the tools available in ANSA and finally meshed. The computational grid is obtained automatically by using the batch tool and exported to the required solver format, ready for the CFD solution.

3. FEM SIMULATIONS AT SALERI

FEM simulations consist of similar phases with respect to CFD analysis. The geometries are firstly cleaned and meshed. The typical components considered in FE-analysis at Saleri are shown in Figure 9.

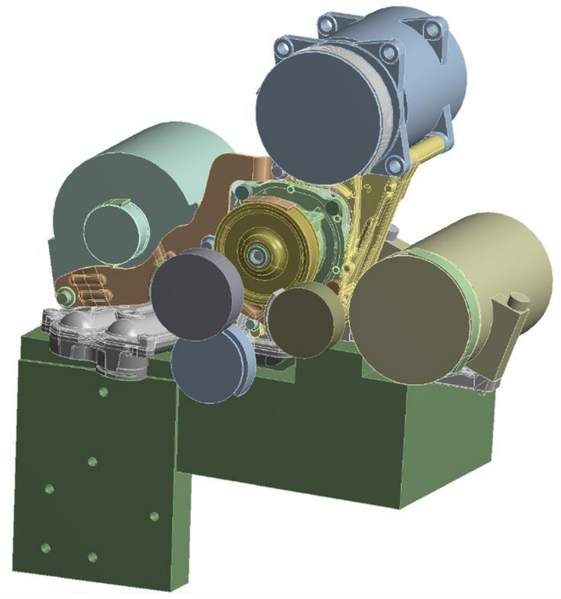

Figure 9 – Typical components considered in FE-analysis at Saleri

The contacts are detected (Figure 10) and exported with the mesh to the FE-solver. Figure 11 shows the displacements of a centrifugal pump subjected to the external loads.

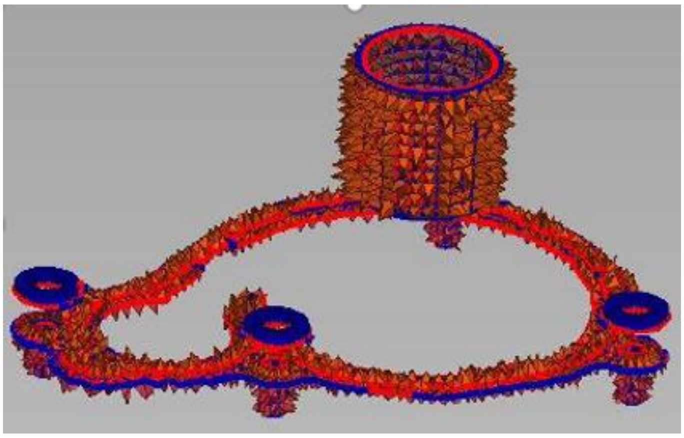

Figure 10 – Contacts detection for the finite element analysis

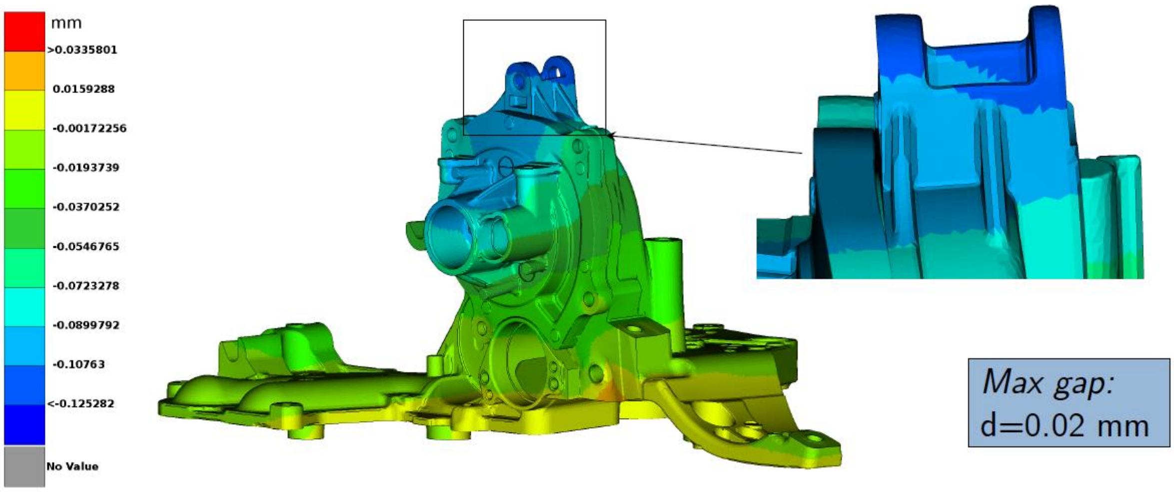

Figure 11 – Displacements of a centrifugal pump subjected to the external loads

The role of BETA CAE software packages

All the pre-processing phases of the FEM simulations described above are performed with ANSA. The contacts detection plays an important role in defining proper and fast simulation setup. The CAD geometry is imported in the software, is cleaned with the tools available in ANSA and finally meshed. The computational grid is automatically obtained by using the batch tool and exported to the required solver format, ready for the FEM solution. The post-processing phase of FE-analysis is performed by using META.

4. MULTIPHYSICS SIMULATIONS AT SALERI

In order to analyse particular phenomena observed in centrifugal pumps, multiphysics simulations need to be performed. Figure 12 shows model and results of a conjugated heat transfer (CHT) simulation performed on an electrical pump for calculating the temperature around the MOSFETs. Figure 13 represents the flow field and stress values as result of a fluid structure interface (FSI) simulation.

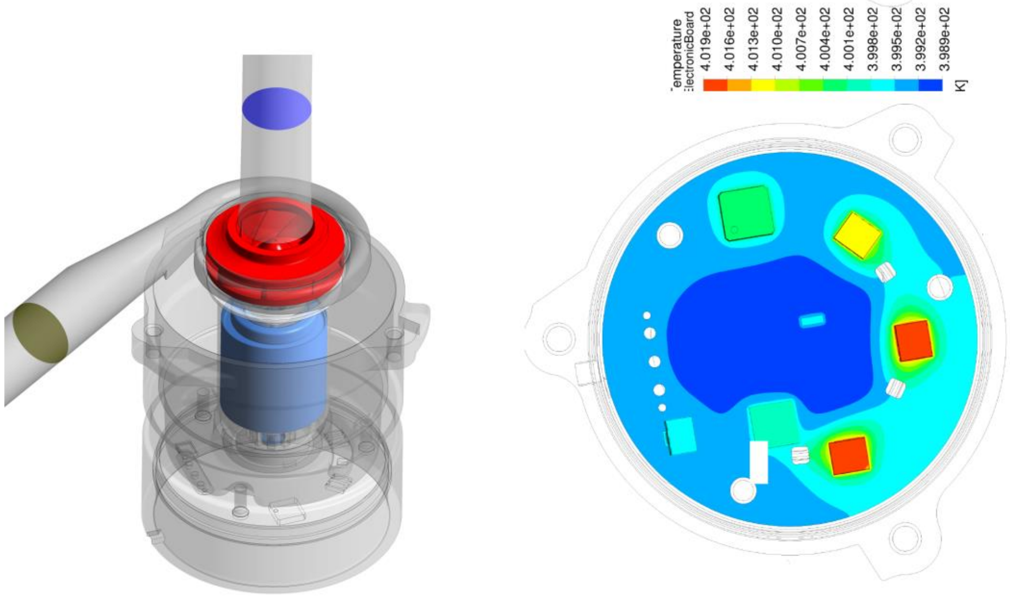

Figure 12 – Model and results of a CHT simulation of an electrical pump

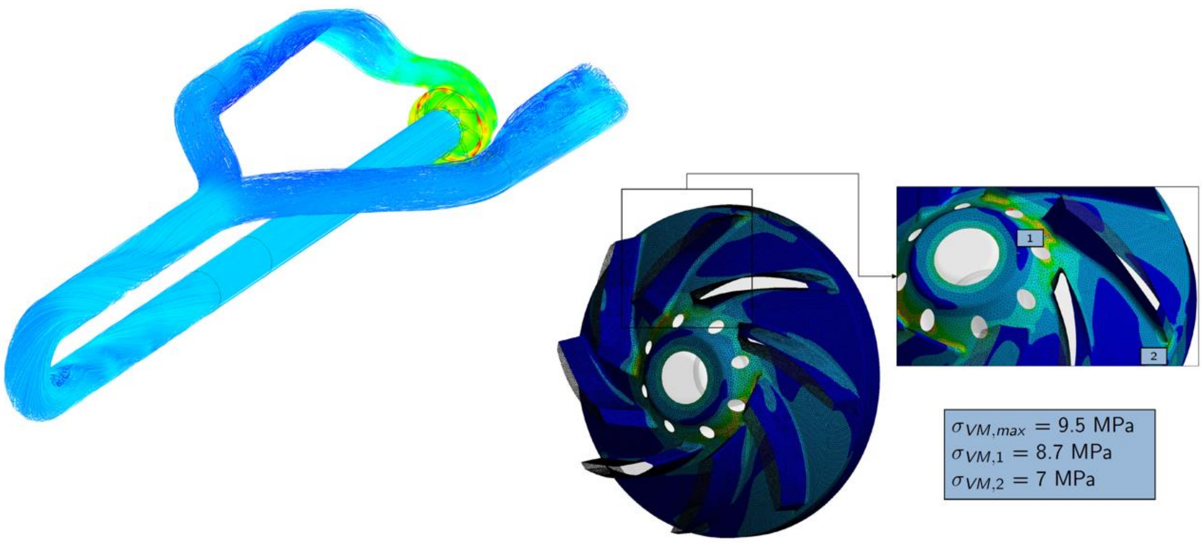

Figure 13 – Flow field and stresses after a FSI simulation

The role of BETA CAE software packages

Also for this kind of simulations the whole pre-processing phase is performed with the ANSA software, in order to clean the geometry and create the computational grid ready for the numerical solution.

5. OPTIMIZATION STRATEGIES AT SALERI

Optimization methodologies are nowadays valuable instruments for meeting high performance designs. Two types of optimizations are performed in Saleri: surrogated-based shape optimization for defining the best hydraulic design of centrifugal pumps (Figure 14) and topology optimization for calculating the best structural shape of the pump housing and bracket (Figure 15).

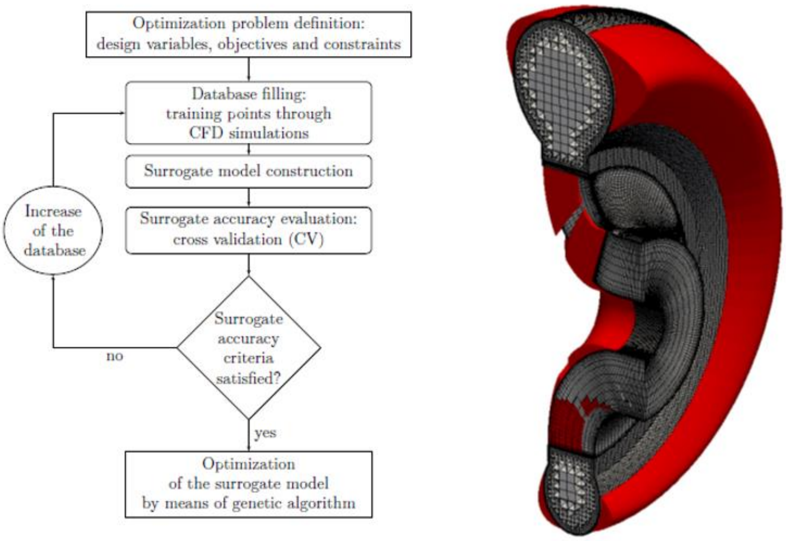

Figure 14 – Surrogate-based shape optimization flowchart (left) and example of results (right)

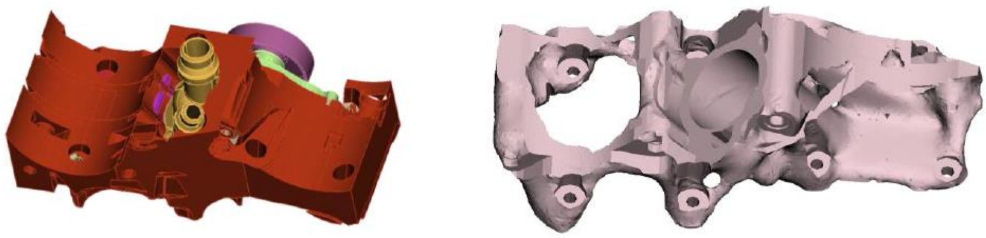

Figure 15 – Topology optimization, initial shape (left) and optimized shape (right)

<u>The role of BETA CAE software packages</u>

The computational grid of the volute in the surrogate-based shape optimization is performed in ANSA, by running a python script in batch mode. The whole topology optimization analysis is performed with BETA CAE tools: the initial geometry is imported, cleaned and meshed with ANSA; the topology optimization is performed with EPILYSIS SOL200 and results are evaluated with META.

6. CONCLUSIONS

The multidisciplinary simulations performed in Saleri have been shown and the use of the BETA CAE software packages has been described. In particular, ANSA has a fundamental role in all the pre-processing phases of simulations at Saleri. EPILYSIS is used for the topology optimization and META is used for the post-processing phase of FE-analysis.
Future works will be devoted to increase the use of META for the post-processing of CFD simulations, by enabling the evaluation of turbo-quantities, such as the pressure coefficient and the skin friction coefficient along the impeller blades at different span positions. Beside the topology optimization analysis of single pump components, EPILYSIS could be used also for the complete pump FE-analysis when the non-linear contacts definition as well as the interference-fit treatment will be available.

REFERENCES

(1) Saleri web page: www.saleri.it