Validation for a Quick and Reliable Procedure for Centrifugal Pumps Using Frozen Rotor Methodology in COMSOL Multiphysics®

Manenti D.[1], Roveglia R.[1], Tanghetti G.[1]
1. Metelli SPA, Via Bonotto 3/5, Cologne (BS), Italy

Introduction Single stage centrifugal pumps are widely used in several engineering fields such as: room conditioning, energetic cycles, automotive industry, home care, etc… Thus, the possibility of simulate their behavior, in terms of pressure increase and mass flow rate, is helpful in reducing prototyping costs in the first design stages.

In the present study the fluid dynamics performances at different regimes of a centrifugal pump for automotive applications (Figure 1) are evaluated using the new COMSOL Multiphysics® Frozen Rotor methodology. The Frozen Rotor allows to take into account inertial terms and Coriolis accelerations even though performing stationary studies, avoiding time dependent analysis with moving mesh and consequently reducing simulation times.

Figure 1. The centrifugal pump for automotive application analyzed

In order to meet industrial needs a quick (low computational cost) and reliable procedure using frozen rotor has been validated giving acceptable comparison with experimental data (Figure 1). For industrial purposes the overall error you commit in a numerical analysis may not be important, but it is fundamental for the error, estimated comparing numerical to experimental data, to be always of the same amount and sign in order to have a reliable comparison between design solutions.

Model Setup The first step of a good numerical simulation is a suitable CAD model. Geometries must be cleaned by all unnecessary features, that are considered useless (see ''the shaft'' in Figure 2.b) and/or harmful for the quality of mesh and solution. An inward and an outward channel are also added to the domain, allowing us to confidently assign mean boundary conditions.

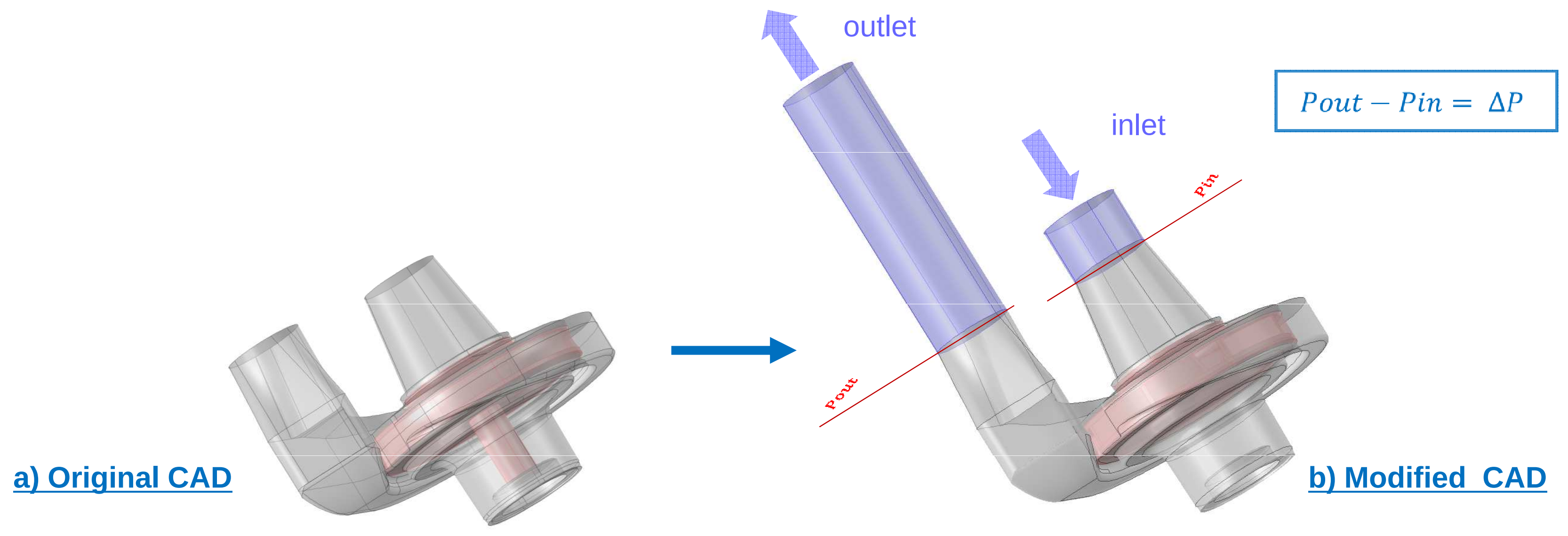

Figure 2. Pump geometry: a) imported as it is, and b) cleaned with COMSOL geometric features

Virtual operations (e.g. "Form Composite Faces") allowed us to simplify surfaces ignoring edges among adjacent surfaces and removing narrow regions in a fast and simple way. Therefore it has been possible to generate homogeneous meshes (see Figure 3) with Free Tetrahedral elements, constant size and grow rate equal to unit for both volute (fixed) and impeller ("frozen" rotating) domains (the built in feature "Swept" is used to mesh the inlet and outward channels).

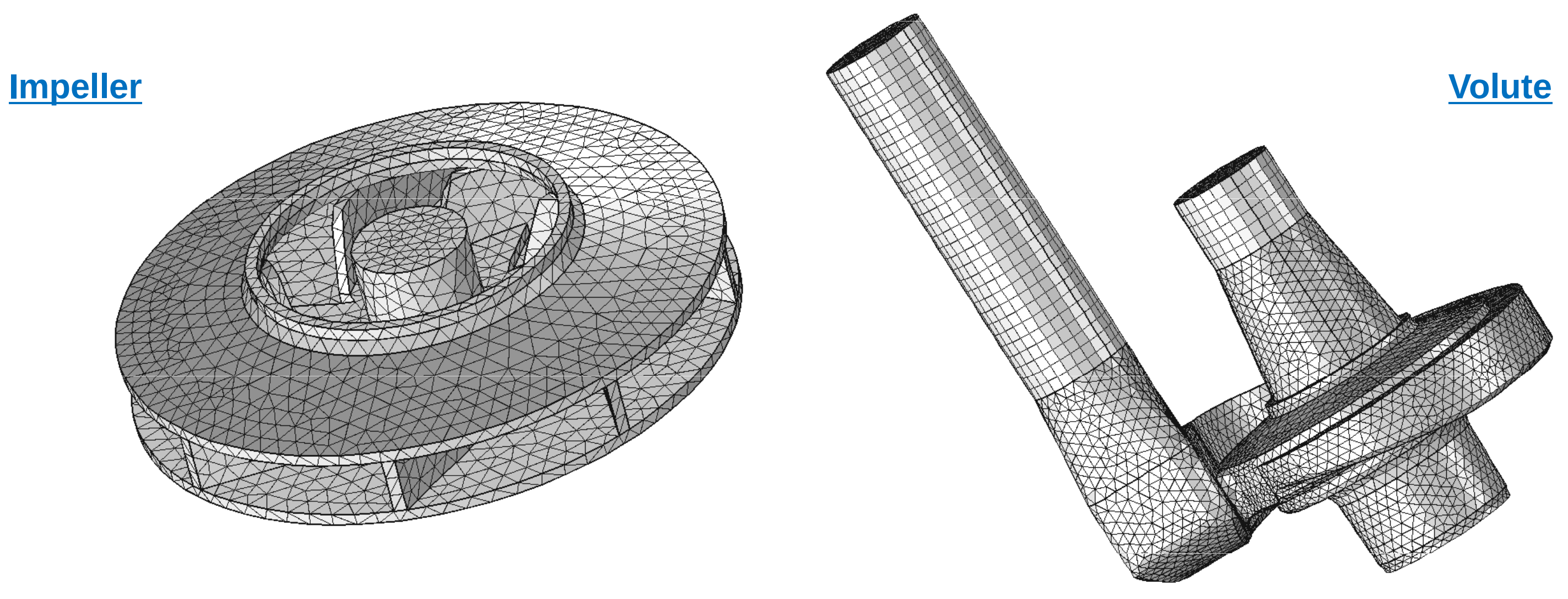

Figure 3. Mesh details

The physics interface Rotating Machinery is used which automatically implements the Frozen Rotor method in computing the flow field. The fluid flow is considered incompressible (50% water and 50% glycol in volume) and a RANS, k-ε (turbulent, wall function) model is used. A normal inflow velocity at inlet and a static pressure at outlet, considered robust boundary conditions have been used . The velocity of the rotating domain is assigned as an angular velocity ω.

A preliminary mesh sensitivity analysis has been performed at 6000 rpm and 3.3 kg/s. In Figure 4., two of the used meshes are shown, the coarsest one is composed by 75k elements (100k DOF, mesh size 5 mm) and the finest by 1336k (1617k DOF, mesh size 1.5 mm). All meshes are realized without boundary layer (BL) and using a constant parameter for element size and a grow rate equal to unit. The results of the analysis are displayed in Figure 4 and indicate that the difference between the convergence value and the coarsest mesh is at most 6%.

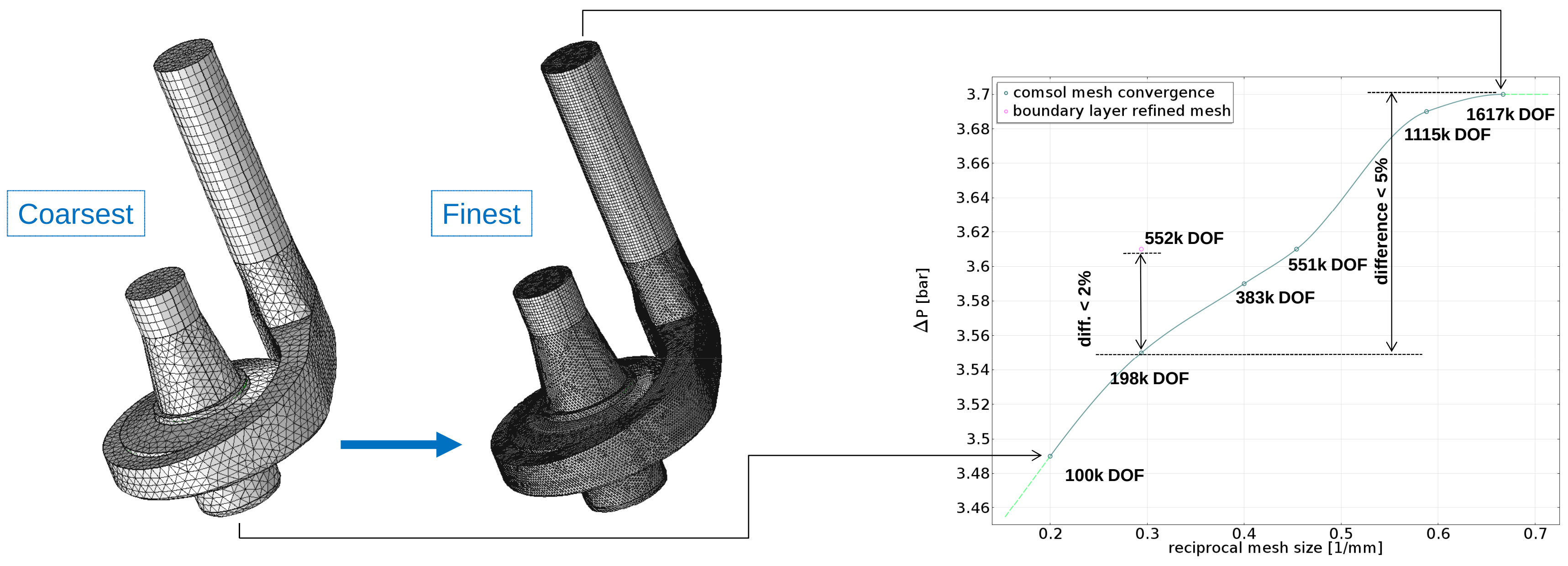

Figure 4. Mesh convergence @ 6000 rpm, 3.3 kg/s

Since our goal is to obtain a quick and reliable procedure, we consider the 156k elements (and 198k DOF, mesh size 3.4 mm) mesh to be a good compromise between the quality of the results and the computational cost, deeming acceptable an error inferior to 5% which accounts for mesh quality. To verify the effect of wall refinement, the same mesh was successively modified introducing boundary layer elements in order to obtain $\delta_W^+ < 11.06$ and a smooth transition from BL mesh to the inner domain one. This refinement increased the number of DOF from 198k to 552k with a difference in results below 2%, which can be considered negligible if compared with the rise in computational cost.

Results To carry out the comparison between experimental and numerical data, a parametric sweep of angular velocity is performed on the 156k elements mesh (without BL). An interpolation function is defined, which has rpm as argument and returns the corresponding mass flow rate used to compute inlet normal velocity. The rpm is then used as a parameter for an automatic study extension that sweeps all evaluation points, which allows us to perform one simulation only.

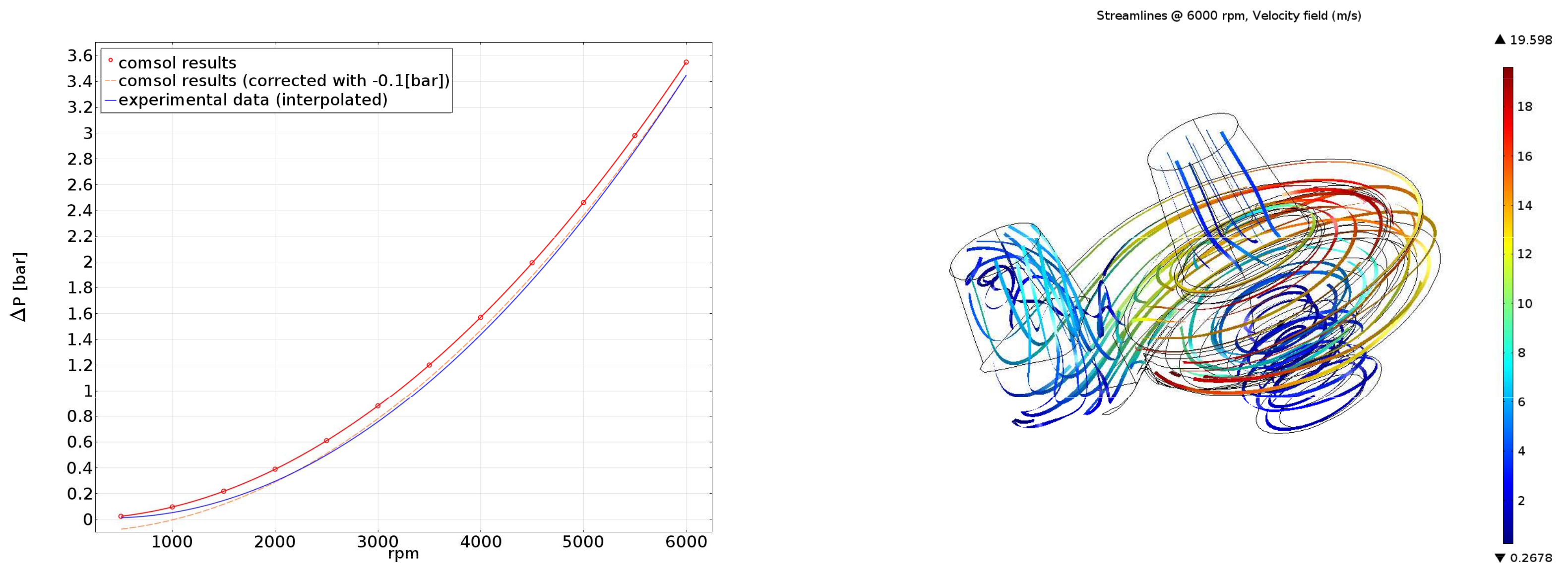

Figure 5. Results

The results in Figure 5 show a good agreement with experiments. The global overestimation in performance is around 0.1 bar except at very low off-design regimes (the curve trends are similar over 2000 rpm). At design conditions the error is less than 5%.

Conclusions The comparison of FEM results with experimental ones is good, underling the suitability of the new Frozen Rotor Methodology. Under an industrial point of view the ratio between computational cost and results quality is satisfying. Hence this validation test case allows us to use COMSOL Multiphysics® for future improvements based on comparison among different designs.

Excerpt from the Proceedings of the 2013 COMSOL Conference in Rotterdam