Development of an innovative non-clogging seal chamber for submersible propeller pumps

Alessandro Corsini[1], Stefano Minotti[1], Franco Rispoli[1], Simone Piccinini[2], Guerriero Romani[2]

[1]*Dipartimento di Meccanica e Aeronautica, Sapienza University of Rome, Italy*
E-mail: *alessandro.corsini@uniroma1.it, minotti@dma.ing.uniroma1.it, franco.rispoli@uniroma1.it*

[2]*Faggiolati Pumps Spa, Italy*
E-mail: *p.simone@faggiolatipumps.it, r.guerriero@faggiolatipumps.it*

Keywords: submersible propeller pumps, non-clogging seal chambers, design methods, virtual prototyping.

SUMMARY. This paper presents the development of a novel non-clogging impeller for a submersible propeller pump. The new design has been obtained in the ambit of an industry-academia collaborative R&D experience, that has benefited from the use of CFD aided engineering to enable the detection of clogging mechanism. The methodology makes use of virtual-prototyping techniques to characterise the hydrodynamic behaviour of the submersible pumps and to design anti-clogging devices. The validity of the virtual prototype (VP) is confirmed by the in-service experience gather by Real Prototype installations in a range of waste water treatment plant configurations.

1 INTRODUCTION

Pumping stations serving cities or industrial districts often comprise one or more electrically driven submersible pumps, generally equipped with diagnosis systems with a high level of automation [1]. This circumstance, as well as the changes in the nature of the sewage and waste water themselves, lead to operational problems related often to pump clogging in the impeller or in the seal chamber [2]. Clogging, or ‘ragging’, remains one of the major challenge in the design or the operation of waste water treatment pumping stations, as such can be very costly in terms of downtime and possible pollution. The severity of such a problem, because of the maintenance complexity, is far more important for the class of submersible pumps that has established as the state-of-the-art solution for waste water applications [3].

Fibres, in the form of household refuses, hygiene articles and diverse industrial wastes, tend to collect in the gap between the impeller and the casing, along the leading edge, at the impeller eye or on the mechanical seals of the impeller. So, though modern wastewater pumps are very reliable, their availability can be impaired by an impeller design that was not chosen to fit the specific application and composition of the wastewater.

When taking into consideration the life cycle costs of wastewater pumps, as one of the selection criteria, it is worth noting that the benchmark values based on the cost of energy and the operation mode are referred to non-clogging, trouble-free wastewater pumps [2]. In presence of clogging, the cost of any remedial intervention, including the consequential costs of pump failure, is the sole decisive cost factor and can easily add up to more than the acquisition value of the pump. Hence, wastewater pump station operators attach maximum importance to operating reliability, and efficiency is only their second decision-making criterion.

Against this background, significant changes are taking place in the design methodology of submersible pumps. Although the conventional approach to industrial design has historically involved trial-and-error empirical methods that rely on the designer's experience of hydrodynamics [4]. Recently approaches to the design of state-of-the-art industrial pumps have utilised computational fluid dynamics (CFD) analyses at the beginning of the design process [5]. Examples of the application of such improved design strategy could be found in the ambit of industrial fan design. Vad [6] and Corsini et al. [7] developed a family of high-performance swept fans for mine ventilation by feeding-back the three-dimensional (3-D) design criterion with computed aerodynamic data about rotor secondary flows. Lee et al. [8] recently applied an inverse approach to the design of cooling fans for electronic appliances, which included the combined use of a 'design of experiments' (DOE) step and computational fluid dynamics (CFD) to explore the space available for design solutions. In doing so, Lee et al. [8] transferred methodologies that had been originally developed for the design of turbo-machinery into the field of industrial fan design. These design methodologies are reliant on CFD to develop appropriate 3-D blade sections [9]. The designers of waste water pumps have historically been primarily constrained by hydrodynamic considerations. An hydrodynamic design would initially be produced by scaling the characteristics of smaller units, with the actual performance of the final design then being established by experimental testing. However, it is no longer acceptable to undertake an hydrodynamic design without considering the implications of it on the pump reliability in waste water handling during the design process.

Such a new design methodology has been addressed in the program of work reported in this paper. The approach was based on CFD tools to provide a *virtual prototyping* design methodology that replaces traditional methods of test and evaluation in pump development. Although virtual-prototyping techniques are presently uncommon in the industrial fan industry, such techniques have been used extensively in other industries. They provide cross-functional evaluations of competing objectives and enable issues that have previously been considered 'downstream issues' to be considered in the initial stages of the design cycle [10, 11]. By developing virtual prototypes (VPs) as 'digital mock-ups', the process of virtual prototyping reduces the need to build physical prototypes and facilitates the early identification of design problems, thus reducing the costs of product development. As shown in Figure 1, the fundamental engineering phase could be complemented by the use of virtual-prototyping activities to characterise the hydrodynamic of the pump impeller and of its sealing flows.

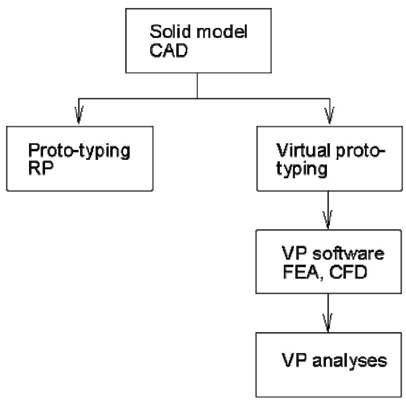


Figure 1: Proto-typing approaches in turbomachinery design

The research project here reported was conducted in the ambit of a collaborative research effort between Faggiolati Pumps Spa and the Fluid Machinery Research Group at Sapienza University of Rome [12, 13]. The project focused on cutting the clogging of propeller pumps used as mixers in waste water ponds and described in Section 2. The CFD tool used to build-up the pump impeller virtual-prototype is summarized in Section 3. Section 4 describes the numerical test program carried on the virtual prototype of the impeller in its original and improved configurations [13]. This section presents also the results of the detailed investigation aimed at the dissection of clogging mechanisms and at proposing hydrodynamic solutions for its passive control. The paper concludes with a summary of the duty data collected from the validation campaigns carried out on the Real Prototypes (RPs) of the developed improved impeller for a range of waste water compositions.

2 SUBMERSIBLE PROPELLER PUMP RANGE

The submersible propeller pumps are used in many applications, like drainage, sewage pumping, general industrial pumping and slurry pumping. The submersible propeller pump investigated belongs to the mixer series GM30 [14], designed for the pumping of wastewater in industrial and civil installations, pumping of mud and of light agricultural sewage. Submersible mixers are used for homogenisation of heavy sludge or liquids with high solid contents, for removal of sedimentary deposits and for to avoid ice formation. A sketch of the GM30 mixer is given in Figure 2.

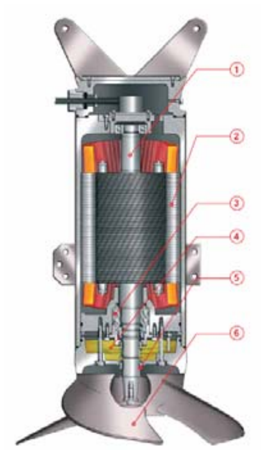
Figure 2: GM30 mixer pump [14]

The GM family of mixers has watertight electric motors connected, by shafts of reduced length. The three-phase asynchronous electric motors, squirrel cage type, have IP 68 protection with class H insulation. They are designed for S1 (continuous) service, with a max overloading up to 10% environmental cooling at temperature < 40ºC. Figure 3 describes a typical mixer configuration and the main components.

In Table 1 the main rating parameters of GM30 mixer are given.

Table 1: Large GM30 submersible mixer description [14]

Type		*GM30B610R1-4T6KA2*
Propeller power	[*kW*]	2.5
Propeller diameter	[*mm*]	300
Capacity	[*l/s*]	210
Thrust	[*N*]	320
Motor type		M610T/M
Rotation speed	[*rpm*]	950
Nominal motor power	[*kW*]	3.4
Max. current	[*A*]	8.5
Starting current	[*A*]	40



1) Shaft stainless steel AISI 316L
2) Electric motor
3) Bearings life lubricated, maintenance free
4) Oil chamber with a ceramic/graphite upper seal
5) Lower seal: silicon carbide/silicon carbide/viton
6) High efficiency propeller casted in AISI 316.

Figure 3: Typical GM mixer pump configuration

Figure 4 provides the head/volume flow rate characteristic curves of the mixer family under investigation.

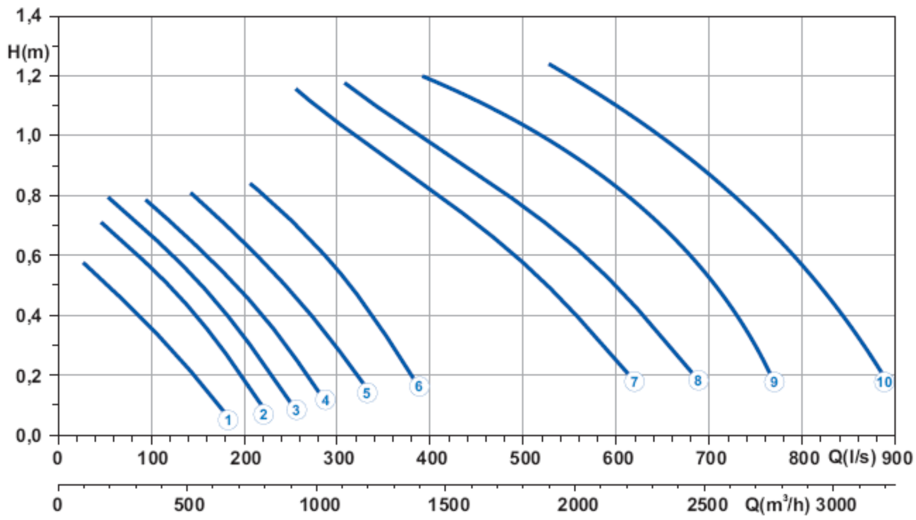


Figure 4: Characteristic curves of the GM mixer range [14]
(1-2: GM30; 3: GM37; 4-6: GM40; 7-10: GM60)

3 VIRTUAL PROTOTYPING TOOLS

3.1 *CFD solver*

The study adopted a parallel multi-grid (MG) scheme developed for an in-house finite-element method (FEM) code [15, 16]. The FEM formulation was based on a highly accurate stabilised Petrov-Galerkin (PG) scheme that had been developed for turbo-machinery applications [17 - 19].

The original parallel solver [16] had been up-graded on the basis of C++ object-oriented technology using ‘libMesh’ library software [20], which facilitates serial and parallel simulation of multi-scale applications using adaptive mesh refinement and coarsening strategies. The solver was designed for built-in integration with software libraries for multi-processor computation portability on Linux-based hardware platforms.

Previous studies, carried out using the current numerical method have shown fair predicting capabilities of the flow physics pertinent to turbomachinery configurations [19, 21].

3.2 *GM30 mixer VP description*

The mixer VP was based on the modelling of: (i) a 120° sector of the seal chamber flow region; and (ii) the set of boundary conditions. The numerical grid of the impeller seal chamber sector consisted of an orthogonal body-fitted coordinate system obtained by the rigid rotation about the pump axis of a block-structured two-dimensional mesh. The overall number of nodes is 226432 nodes, i.e. 3712 on the meridional section and 61 in the circumferential direction, resulting in 222660 hexahedral elements. Figure 5 shows a perspective view of the computational domain.

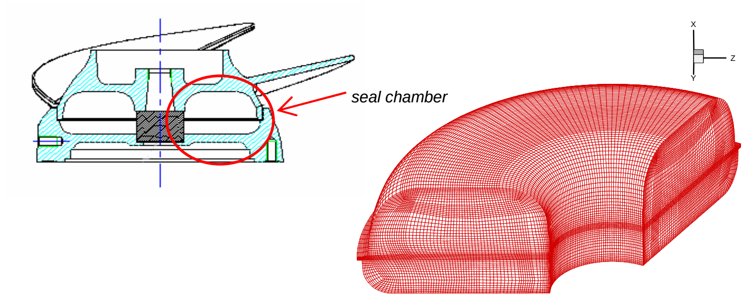


Figure 5: Computational mesh

The peripheral gap, establishing the sealing of the chamber, was modelled by using 2135 nodes.

As far as the boundary data were concerned, the set of conditions is summarized in Figure 6.

Boundary surfaces	*u*	*v*	*w*
α–β–ϕ–ε	0	$\omega r\gamma cos(\theta)$	$\omega r\gamma sin(\theta)$
a-b-f-e	0	0	0
b-β–ϕ–*f*	0	0	0

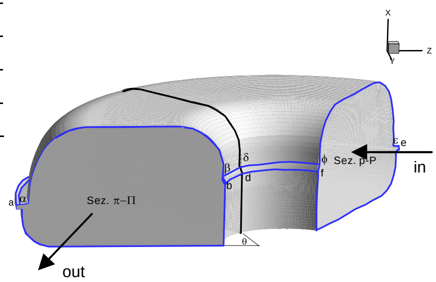


Figure 6: Boundary condition modelling

It is worth noting that the upper surface of the chamber moves with the impeller producing a solid body rotation velocity distribution on the boundary nodes. Moreover, along the permeable annular surface a-α-ε-e, a circumferentially non-uniform pressure distribution was used in order to simulate the pressure drop due to the coupling between the impeller and the fluid, taking place on the pressure and the suction surfaces of the blade. On the inlet and outlet sections, respectively section p-P and section π-Π (Figure 6), the periodicity condition was used only on the velocity components.

4 SEAL CHAMBER HYDRODYNAMIC INVESTIGATIONS

4.1 *Original configuration CFD analysis*

The simulation of the GM30 seal chamber flow field was motivated by the need of detecting the hydrodynamic mechanisms at play in the transport of fibrous materials radially across the chamber, i.e. from the annular gap inward to the inner mechanical seal.

To this end, the numerical campaigns were run under the hypothesis of negligible interaction between the dispersed fibers and the water fluid dynamics.

Figure 7 shows the evolution of the flow field according to the circumferential direction. Static pressure contours and streamlines were shown on three meridional sections, namely at a) $\theta = 0°$, b) $\theta = 60°$, and c) $\theta = 110°$.

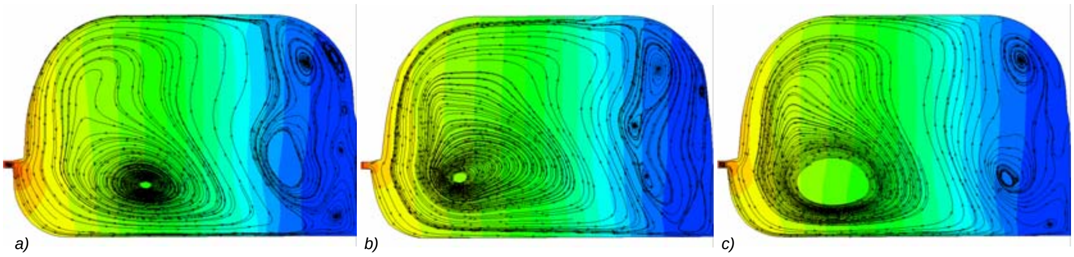


Figure 7: Circumferential evolution of static pressure and streamlines, a) at $\theta = 0°$, b) $\theta = 60°$, and c) $\theta = 110°$.

As given by the seal chamber flow field tomography, Figure 7, it is apparent that the hydrodynamics of waste water is mainly driven by the rotation of the upper surface. As a consequence of the differential Couette-like conditions established along the radial direction, owing to both the surface velocity and the chamber height, the flow field features a large anti-clockwise outer vortex inducing inward material movement on the fixed wall of the chamber. In proximity of the impeller shaft, moreover, was discernible a set of vortical patterns those originating in the chamber upper and lower corners and rolling in the region of minimum static pressure.

4.2 *Improved configuration CFD analysis*

The detection, by means of the pump VP, of the mechanism governing the passive transport throughout the chamber and potentially responsible for the seal clogging allow for the definition of design correlation between the chamber geometry and any target static pressure radial distribution [12].

The application of such a correlation resulted in a set of improved geometries based on the variation of the seal chamber height [13]. The improved design was intended as a passive control system for the pump clogging.

A typical member of the space of the design solution is described in Figure 8 by showing a meridional of the propeller pump impeller and seal chamber.

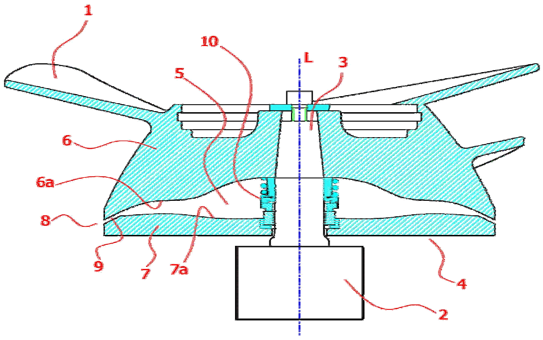


Figure 8: GM30 impeller and seal chamber improved geometry [13]

In order to demonstrate the hydrodynamic pay-off given by the improved impeller design, Figure 9 describes the static pressure field and the streamlines respectively on a) a probing plane orthogonal to the pump axis in proximity of the chamber fixed wall, and b) a meridional section of the chamber.

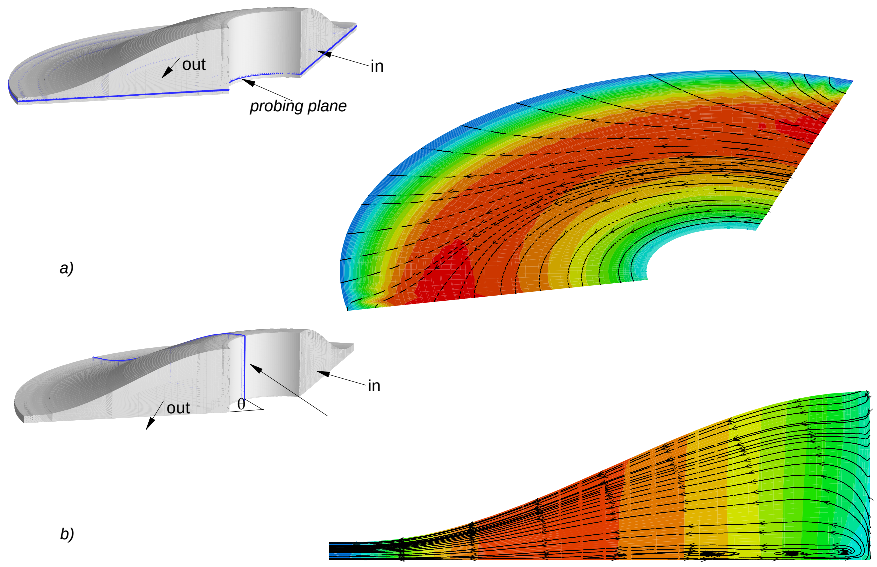


Figure 9: Static pressure coefficient map in the seal chamber on a) a section orthogonal to the impeller axis, and b) a meridional section.

The analysis of the pressure field demonstrates that the pressure peak was shifted toward inner radii and as such it was able to establish a sort of pressure potential which contributed to the chamber sealing on the outer annulus.

Furthermore, when looking at the stream-paths it is apparent that the geometrical modification of the chamber wall distance resulted in the destruction of any organized vortical structure reducing the chance for the fibrous materials to be transported radially inward through the chamber volume.

5 SUBMERSIBLE PROPELLER PUMP *RPs* DUTY DATA

The assessment of hydrodynamic pay-off, carried out by using the impeller VP, the validation phase of the new design has been followed by an on-site testing campaigns. This campaigns, summarized in Table 2, has been carried out running a number of RP of the improved impeller in waste water treatment plant with a variety of waste water contaminants and for different pump duty conditions.

Table 2: Summary of improved impeller RPs duty data

Waste water plant	**Types of waste water**	**Waste water basin volume**	**RP duty type**	**Original time between failure**	**RP duty time**	**RP seal chamber status**
Faggiolati Pumps Spa Lab	Clean water and selected pollutants: - *straw* - *hair* - *sand*	3 m × 5 m ×1,7 m	continuous	NA	5 days	non-clogged
Feedstock plant	Agricultural waste water - *high fiber content*	4 m × 4 m × 6 m	continuous	clogged in 7-10 days	6 months	non-clogged
Municipal plant	Civil waste water - *medium fiber content* - *sand*	8 m × 20m × 6 m	continuous	clogged in 1 month	5 months	non-clogged
Paper-mill plant	Paper pulp - *glue content*	3 m × 4 m × 3 m	intermittent	clogged in 7-10 days	7 days	clogged

As given by the data in Table 2, the validity of the CFD-based design has been largely confirmed by the in-service experience in real duty conditions. The performance improvement is demonstrated by the lack of any clogging during duty time largely longer than the original time between failure. While the unique unsuccessful application is related to special fluid handling, such as the paper pulp contaminated by glutinous component.

The definition of an improved non-clogging impeller configuration for the GM family of mixers led to the formulation of an international patent recently granted [13].

6 CONCLUSIONS

This paper has presented a novel design to prevent the clogging phenomenon in submersible propeller pumps used as mixer in waste water treatment plants.

The new approach implemented features: (i) an up-front role for computational tools (such as CFD); and (ii) an enhanced role for virtual prototyping as opposed to traditional test-and-evaluation development. The improved design is based on the interpretation of the hydrodynamic mechanisms responsible for the transport through the seal of fibrous and solid dispersed materials. The developed seal chamber design demonstrated, on VP and RP validation campaigns, to exploit a passive control of the seal clogging based on the idea of controlling the hydrodynamic pressure field in the impeller seal chamber.

The new design methodology has facilitated the development of a new range of propeller pumps comply with the requirements of extending failure-free operation, irrespectively from the waste water compositions, in a number of industrial and municipal plants.

ACKNOWLEDGEMENTS

The present research was done in the context of the contract Faggiolati Pumps-DMA04. The authors gratefully acknowledge Dr. A. Santoriello for his contribution in the earlier stage of the work.

References

[1] Thamsen, P.U., "Cutting clogging in wastewater", *World Pumps*, **511**, 22-25 (2009).

[2] Bross, S., Pensler, T., and Springer, P., "Effects of impeller geometry on reliability and costs of wastewater pumps", *World Pumps*, **452**, 34-38 (2004).

[3] Berezin, S., "Submersible pumps for wastewater applications", *World Pumps*, **480**, 26-30 (2006).

[4] Stepanoff, A.J., *Centrifugal and Axial Flow Pumps: Theory, Design, and Application*, Wiley, New York (1948).

[5] Spann, J., and Horgan, S., "Cutting clogging in wastewater", *World Pumps*, **480**, 32-35 (2006).

[6] Vad, J., "Incorporation of Forward Blade Sweep in the Non-Free Vortex Design Method of Axial Flow Turbomachinery Rotors", *Periodica Polytechnica, Mechanical Engineering Series*, **45-2**, 217-237 (2001).

[7] Corsini A., Rispoli F., Vad J. and Bencze F., "Non free vortex flow effects in an axial flow rotor", *Periodica Polytechnica, Mechanical Engineering Series*, **45-2**, 201-216 (2001).

[8] Lee, K.Y., Choi, Y.S., Kim Y.L. and Yun J.H., "Design of axial fan using inverse design method", *J. of Mech. Science and Tech.*, **22**, 1883-1888 (2008).

[9] Horlock, J.H. and Denton, J.D, "A review of some early design practice using computational fluid dynamics and a current perspective", *ASME J. Turbomachinery*, **127**, 5-13 (2005).

[10] Pratt, M.J., "Virtual prototypes and product models in mechanical engineering", *IFIP WG 5.10 on Virtual Environments and their Applications and Virtual Prototyping*, 113-128 (1994).

[11] Jasnoch, U., Kress, H., and Rix, J., "Towards a virtual prototyping environment", *IFIP WG 5.10 on Virtual Environments and their Applications and Virtual Prototyping*, 173-183 (1994).

[12] Corsini, A., Rispoli, F., and Santoriello, A., RR1&2/FAGGIOLATI 2004 - *Dipartimento di Meccanica e Aeronautica*, Università di Roma "La Sapienza" (2005).

[13] Corsini, A., Rispoli, F., Romani, G., and Piccinini, S., "Submersible mixer for wastewater treatment", Faggiolati Pumps Spa., International patent application No. MC2007 A 000152 (2007).

[14] Faggiolati Pumps Spa, *Product Catalogue* (2009).

[15] Borello, D., Corsini, A., and Rispoli, F., "A parallel approach to FEM modelling of internal, fluid flows", *5th European SGI/Cray MPP Workshop - CINECA*, Bologna, Italy (1999).

[16] Borello, D., Corsini, A., and Rispoli, F., "A finite element overlapping scheme for turbo-machinery flows on parallel platforms", *Computers and Fluids*, **32-7**, 1017-1047 (2003).

[17] Corsini A., Rispoli, F., Santoriello, A., and Tezduyar, T.E., "Improved discontinuity-capturing finite element techniques for reaction effects in turbulence computation", *Computational Mechanics*, **38**, 356-364 (2006).

[18] Rispoli, F., Corsini A., and Tezduyar, T.E., "Finite element computation of turbulent flows with the Discontinuity-Capturing Directional Dissipation (DCDD)", *Computers & Fluids*, **36**, 121-126 (2007).

[19] Corsini, A., Menichini, F., Rispoli, F., Santoriello, A., Tezduyar, T.E., "A multiscale finite element formulation with discontinuity capturing for turbulence models with dominant reactionlike terms", *Journal of Applied Mechanics*, **76** 021211 (2009).

[20] Kirk, B., Peterson, J.W., Stogner, R.H., and Carey, G.F., 2006, "LibMesh: A C++ Library for Parallel Adaptive Mesh Refinement/ Coarsening Simulations", *Engineering with Computers*, **22**(**3-4**), 237-254 (2006).

[21] Corsini, A., Minotti, S., Sciulli, F., and Sheard, A.G., "The role of computational methods in the development of an aero-acoustic design methodology. Part I: Application to a family of large industrial fans", *CMFF09*, Budapest, Hungary, September (2009).