CFD OPTIMIZATION OF A FAN FOR INDUSTRIAL APPLICATIONS

Nicola ALDI[1], Carlo BURATTO[1], Alessandro CARANDINA[1,2], Michele PINELLI[1], Alessio SUMAN[1], Andrea ZANARDI[3]

[1] *Engineering Department in Ferrara (EnDiF), via Saragat 1, 44122, Ferrara, Italy*

[2] *Fluid-A s.r.l., via Benedetto Zallone 19, 40066 Pieve di Cento (BO), Italy*

[3] *MZ Aspiratori S.p.A., via Certani 7, 40045 Budrio (BO), Italy*

SUMMARY

Energy consumption and operating ranges in industrial applications of fans have grown in importance in the recent years and now, in order to be attractive for the market, the product must fulfill a number of requirements such as: (i) good efficiency inside all the operative range, (ii) higher maximum flow rate and (iii) minimum manufacturing and maintenance costs. A CFD study, regarding three industrial centrifugal fans for low, medium and high pressure applications, will be carried out showing how the impact of design optimization on the meridional passage and blade shape changes according to the specific speed. An experimental campaign on a limited number of the modified fans confirms the overall design guidelines deducted from the CFD.

INTRODUCTION

Energy consumption and savings in domestic and industrial applications of fans have recently gained attention in Europe. In a recent European Directive [1], eco-design demands for products that can have a critical environmental impact and, at the same time, can present significant potential for improvement through innovative design, have been set. In most cases installation and process requirements (such as space constraints, electrical power limits and national and international directives in the field of health and safety) do not match with consolidated fluid dynamic design approaches. In this context, Computational Fluid Dynamics (CFD) is the key tool which can complement traditional one-dimensional and bi-dimensional design approaches. The CFD tool can be used for fan optimization in order to improve the fan performance. In this paper fan performances does not only refer to the pressure ratio, mass flow rate and efficiency at the design point (or best efficiency point) but, more in general, will refer to the amplitude of the operating fan region. In fact, some applications, such as air conditioning systems, require very different fan performance during the normal operation. In recent years, different control strategies (like fuzzy logic, [2]) and control devices (like inverter, [3]) or active/passive devices based on temperature activated Shape Memory Alloys [4] are used in order to optimize the relationship between the process requirements and the energy management. In this context, using a fan with a flat efficiency curve improves the energy optimization of the entire system.

Given this background, fan manufacturers have to manage numerous requirements beyond those related to the energy efficiency, and in particular they have to (i) limit the fan dimension and weight (impeller, casing and electric mover), (ii) guarantee an easy implementation on their existing production line, and (iii) ensure the serviceability of the existing systems and plants that impose dimension and shape of the inlet and outlet section of the fan. Due to these limits and restrictions, the classic one-dimensional methodology for centrifugal fan design [5, 6] must be supported by a three-dimensional methodology tool (like a CFD numerical analysis) that allows i) the in-detail optimization especially in the case of off-design operating conditions [7], ii) the reduction of tested fan prototypes, thus reducing costs and (iii) an improvement of the fan performance thanks to the possibility of analyzing the entire flow field inside the turbomachine. Some examples in this sense are: [7] where an automotive engine cooling system was significantly influenced by the limited space of the engine bay and the ram effect resulting from the vehicle's motion; [8] where Tallgreen investigated the potential of CFD optimization; [9] where an integrated 1D/3D numerical procedure was developed and [10,11] which focused on the internal flow analysis of the centrifugal fans.

In this paper the Ansys CFX CFD package will be used for the optimization of two centrifugal fans used for process industry applications. In this article the term optimization refers to the activity of design and test new solutions (driven by the experience and the knowledge of the designers) in order to achieve the objective of the study. This approach can be conceived like a human driven heuristic optimization but no automatic single/multi-objective optimization study will be carried out on the fan geometries. The two fans refer to (i) a low pressure fan (wider meridional flow passage and lower rotational speed) and (ii) a middle pressure fan (narrower meridional flow passage and higher rotational speed). The two fans are equipped with two different impellers that share the same outer diameter. The optimization presented in this paper shows how the CFD tool can be useful to tailor the shape impeller/casing for each machine and the related specific application requirements.
The optimization of the fan is composed of: (i) a validation of the CFD numerical results with the experimental ones, (ii) the analysis of the actual fan geometry by using the CFD numerical results in order to point out the critical aspects of the analyzed geometries and finally (iii) the analysis of the fan impeller and casing geometry modifications and the evaluation of the fan performance (mass flow rate, pressure ratio and efficiency). The extension of the fan efficiency trend will be the main objective of the fan geometry optimization.

STRATEGY FOR CFD OPTIMIZATION

The aerodynamic design of the centrifugal fans is based on analytical relationships that use the assumption 1D/2D ideal flow and on empirical corrections obtained from statistical data [5]. In this design process the main components of the fan (i.e. inlet cone, impeller and volute) are individually designed and their interactions are neglected [5]. However, the real flow is different from the ideal situation and therefore it is impossible to predict precisely the fan performances at the design stage. The main issue of the aerodynamic design is the 1D/2D assumption. In fact the nature of the flow-field within a centrifugal fan is highly three-dimensional and turbulent, as the flow is turned from an axial to radial direction. Moreover, the effect of interactions between stationary and rotating components on fan performances should be taken into account. In this paper, three-dimensional optimization is carried out through the use of CFD numerical simulations with an empirical approach and on new analytical relationships that use the CFD data. The reference impellers and casing geometries refer to a one-dimensional design provided by the manufacturer. At the beginning of the optimization, the computational domain and the numerical settings are validated against the experimental results.

The numerical simulations are carried out by means of the commercial CFD code ANSYS CFX 14.5. The computational domain is composed of two stationary domains (inlet cone and volute) and one rotating domain (see "rotor" domain in Fig. 1c). To reduce computational effort, only one blade passage is modeled for the impeller and only a section of the same angular extent is modeled for the inlet cone domain. For these reasons a one-to-one node matching periodicity is also used. The grid used in the calculations is a hybrid grid generated by means of ANSYS Meshing 14.5. Prism layers are added on the surface of the blade to help solve the flow around the blade (see Fig. 2a).

The simulations are performed in a steady multiple frame of reference, taking into account the contemporary presence of moving and stationary domains. In particular, a mixing plane model is imposed at the interfaces between rotating and stationary domains and used for all the simulations. Thakur et al. [12] developed and tested two quasi-steady rotor/stator interaction models in order to predict the performance of centrifugal blowers as the mass flow rate is varied. Both models were observed to predict the correct trends in all global measures of performance.

The turbulence model used in the calculations is the standard *k-ω SST* model. Near-wall effects are modeled with an automatic treatment that uses the wall function approach or the low Reynolds number turbulence model standard *k-ω*, depending on the grid refinement in that specific region. The far wall effects are modeled with the *k-ε* turbulence model. The SST model can be applied on arbitrarily fine meshes and it allows a consistent mesh refinement independent of the Reynolds number [13].

Since only a section of the impeller and the inlet cone geometry is modeled, the rotational periodic boundary condition is applied to the lateral surfaces that delimit the section from the adjacent ones. The maximum local Mach number is lower than 0.15 and the incompressible flow hypothesis is adopted. Air at 20 °C (constant) is imposed as a medium and total pressure and flow angle are imposed at the inflow boundary. The inlet total pressure $(p_0)_{in}$ is imposed at 101,325 Pa. The mass flow rate is imposed at the outflow boundary in order to perform the entire performance trend of the fans. The considered rotational speed is that of the design.

A mesh sensitivity analysis has been performed for between four million and ten million elements. Figure 2b reports the mesh sensitivity analysis by using the non-dimensional fan total pressure and shaft power as a function of the total number of mesh elements (N). The total number of mesh elements is the sum of tetrahedral and prism elements. The independence of the results is obtained for approximately eight and a half million elements. Therefore, the final mesh is composed of about 8 million elements with 4 prism layers as reported in detail in Fig. 2a.

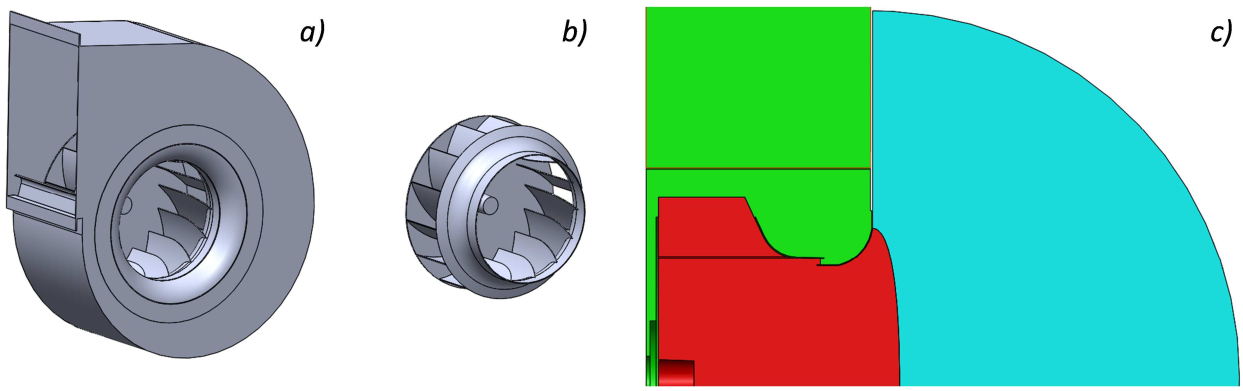


Figure 1: a) LP fan assembly, b) LP fan impeller, c) computational domain of the LP fan, the cyan part represents the inlet cone domain, the red part is the rotor domain and the green part is the volute domain.

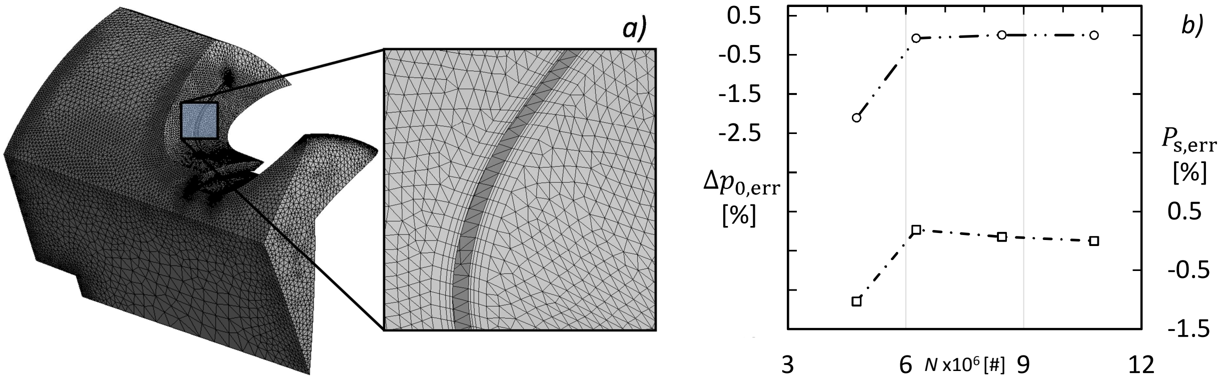


Figure 2: a) computational hybrid mesh and b) grid sensitivity analysis for the LP fan (the chart shows the relative error of the performance with respect to the asymptotic value).

For the sake of brevity, mesh sensitivity analysis and CFD numerical validation are reported only for the LP fan. Trends in Fig. 3 report the comparison between the CFD numerical results and experimental results obtained for the LP fans. The comparison refers to the initial fan geometry, before the CFD optimization. The experimental test was carried out in agreement with the standard EN ISO 5801 while the CFD simulation was carried out in agreement with the numerical model and description reported above. The shape of both the experimental performance maps is correctly reproduced by the numerical code. The non-dimensional total pressure increase trend highlighted that the CFD results overestimate the experimental data but in a very consistent way. The deviations in the comparison of fan efficiency are very low and in some cases, negligible. Thus, the numerical values are in fairly good agreement with the experimental data and since the aim of the validation was to obtain a fan model, the numerical model can be considered reliable.

For the Low Pressure (LP) fan the optimization refers to the modification of the (i) meridional passage width and (ii) mouth passage shape. For the Middle Pressure (MP) fan the optimization refers to the modification of the impeller blade shape through (i) a parametric method and (ii) by coupling analytical method and CFD results.

LP CASE

The fan performance will be scaled with the design point of the LP fan. The η_{DES} of this fan is in line with the European Legislation (327/2011), but to match better with market requests, a higher total pressure increment without increasing the shaft power is needed for higher flow rates. The CFD analysis of the LP fan showed that the main issue of this machine is a big flow separation on the shroud for higher flow rates (see Fig. 4a), mainly due to the increment of passage width from the fan mouth to the leading edge of the blade. The obvious way to optimize the passage is to change the impeller inlet diameter and the shape of the shroud, but since it is manufactured by metal sheet technology, the parameters are chosen to be easy to integrate into the existing production line.

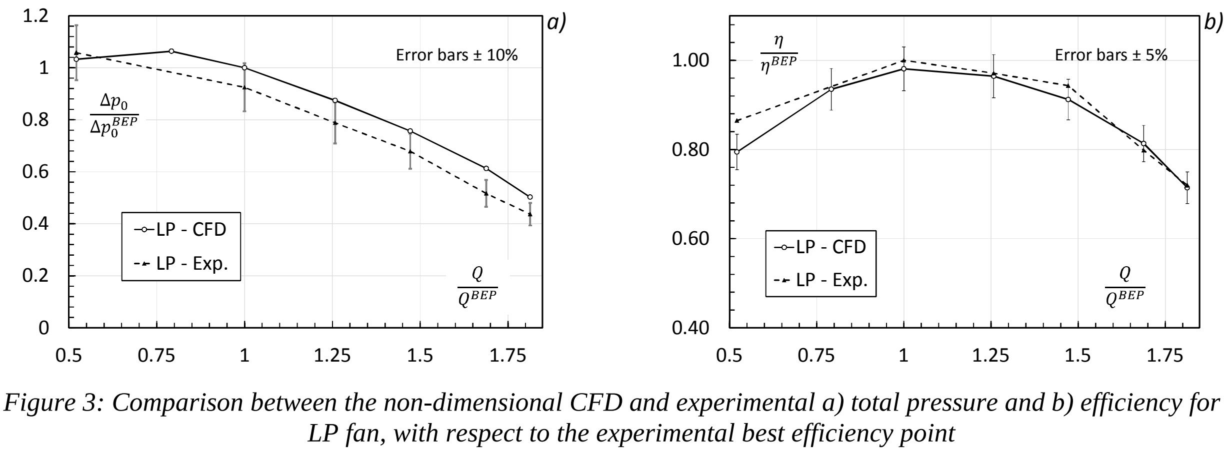
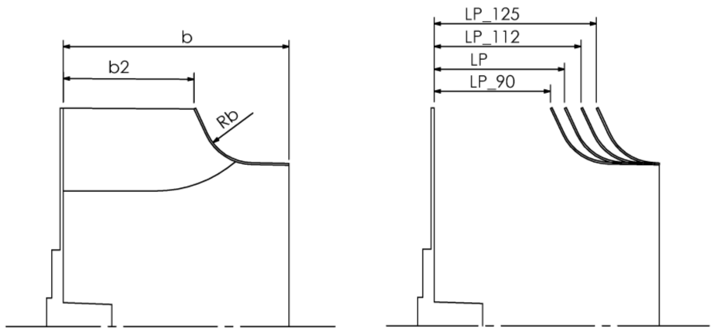


Figure 3: Comparison between the non-dimensional CFD and experimental a) total pressure and b) efficiency for the LP fan, with respect to the experimental best efficiency point



(a) *(b)*

Figure 4: modification of the fan width b2.

The parameters chosen are i) the distance between the hub and the shroud and ii) the shape of the mouth.

Meridional Passage Width

The outlet passage width $b2$ is a key parameter for the performance at BEP but it also influences the shape of the performance maps. Hence a modification of the sole $b2$ parameter should be carefully investigated and evaluated to find the optimal value in order to reduce the flow separation at the shroud and, at the same time, improve the performance at high flow rate. This optimization is not straightforward and even recently CFD studies have been proposed to assess the influence of shroud modification on performance. For instance, in [14] a study from Dresser-Rand investigates the aerodynamic effect associated with varying the shroud curvature and axial length of a centrifugal compressor impeller. In this paper in order to investigate the influence of the passage width $b2$, three different geometries were tested through a CFD analysis.

The passage width modification was created so that the shape of the shroud and of the mouth would be unchanged. This means that the radius Rb and the width b are constant for all the modifications made (see Fig. 4a). Three values of $b2$ were chosen and in particular $b2$ equal to 90 %, 112 % and 125 % of the original fan (LP) width were tested (named LP_90, LP_112 and LP_125, respectively).

As can be seen from Fig. 5a-c, for all the geometries tested the performance at higher flow rates is worse with respect to the original fan. This confirms that from an overall performance point of view the $b2$ parameter is already an optimal one for the LP fan under investigation. From Fig. 6 it can be also seen that the size of the flow separation on the shroud grows with the passage width $b2$.

On the contrary for a $b2$ smaller than the original the flow separation seems to be reduced, but the effect is hindered by the reduction of blade loading due to the narrower passage.

Mouth Passage Shape

In order to reduce the flow separation at the shroud the flow at the impeller inlet should be as radial as possible. The mouths of LP_M1 and LP_M2 fans were conceived to give the flow a radial component before entering the impeller. Both the impellers were designed using an arc of ellipse, but with different orientations. The LP_M1 mouth (see Fig. 7b) has a sudden restriction of the passage, followed by a diffusion of the channel; the straight part is tangent to the shroud in order to let the flow approach it smoothly. The dimensions of the arc of ellipse are chosen according to the experience of the operators. The LP_M2 mouth (see Fig. 7c) also uses the arc of ellipse, but with a different orientation. With this configuration the restriction is more gradual, and the ending is pushing the flow against the shroud; this second shape is also designed similar to a typical Venturi tube for a diameter equal to the eye of the impeller.

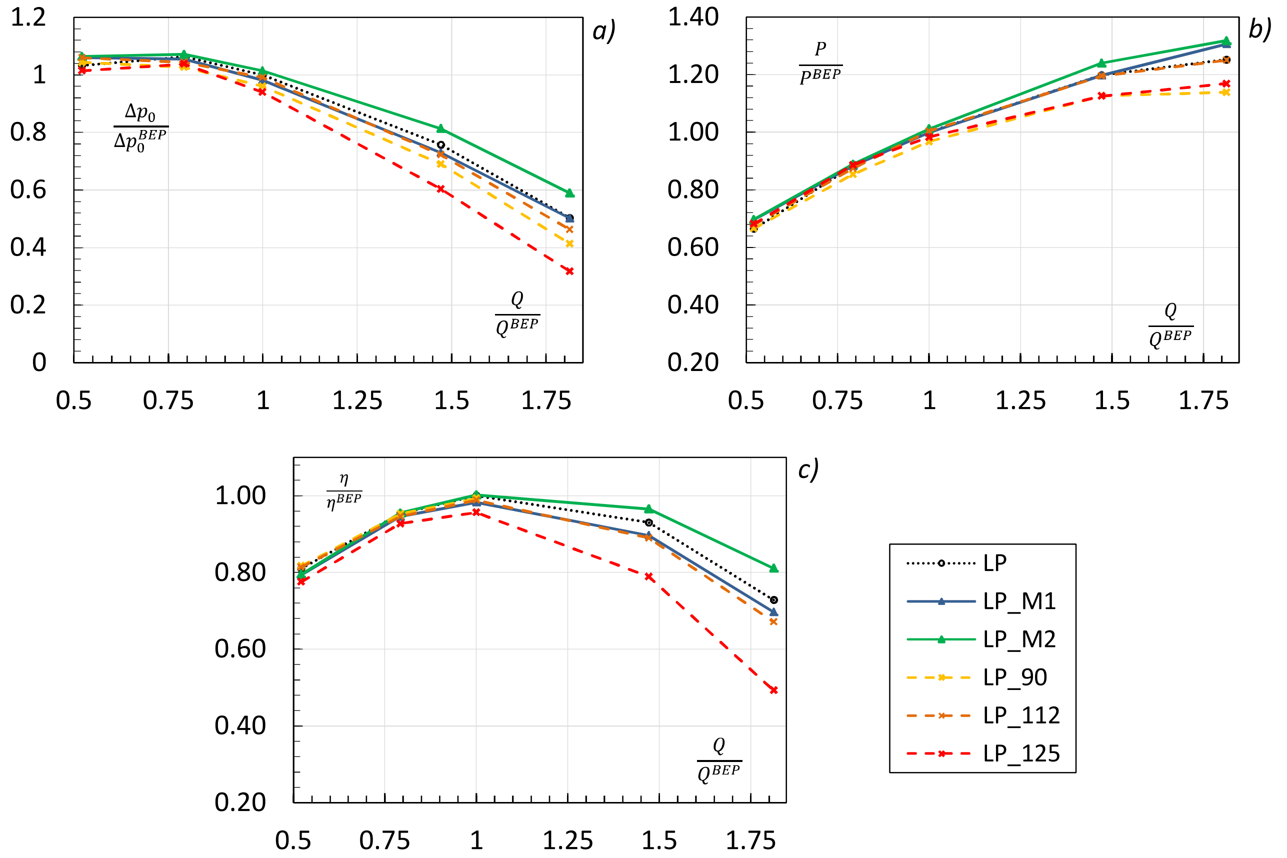
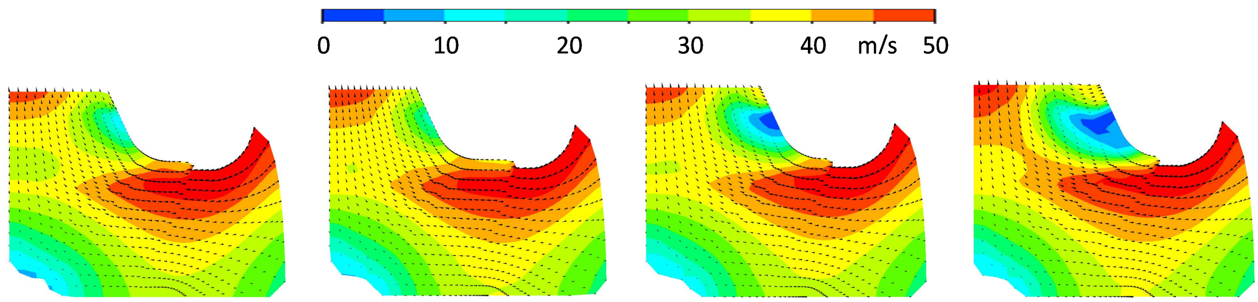
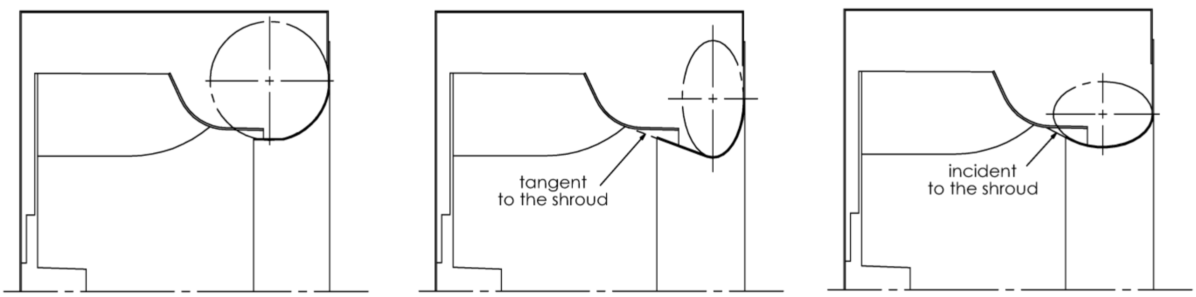


Figure 5: comparison of a) normalized total pressure increment, b) normalized shaft power and c) normalized efficiency for the fans LP, LP_90, LP_112, LP_125. The normalization is done with respect to the Best Efficiency Point calculated with CFD for the LP fan.



(a) (b) (c) (d)

Figure 6: Meridional passage circumferential averaged velocity in the rotating frame of reference, at non-dimensional flow rate Q/Q^{BEP}=1.76, respectively for the fans: a) LP, b) LP_90, c) LP_112, d) LP_125. Q^{BEP} is the Best Efficiency Point flow rate for the LP fan, calculated with CFD.



(a) (b) (c)

Figure 7: shape of the mouth passage of the a) MP, b) MP_M1 and c) MP_M2 fans.

The meridional velocity plot in Fig. 8c shows that the LP_M2 mouth actually reduces the separation on the shroud, considerably improving the efficiency and consequently the total pressure increase at higher flow rates (Fig. 5). The shaft power of the LP_M2 increases slightly.

MP CASE

The η_{DES} of this fan is in line with the European Legislation (327/2011), but to better match with market requests, the range of operation has to be extended so higher total pressure increment and efficiency is needed for higher flow rates. The obvious way to proceed is to redesign the entire impeller (inlet and outlet blade angle/diameter/passage width, number of blades) for a different design point, at a higher flow rate, but this would be expensive and would probably lead to the decay of the performance at the actual design point, so the solution proposed consists of i) the modification and ii) the redesign of the actual MP impeller blade.

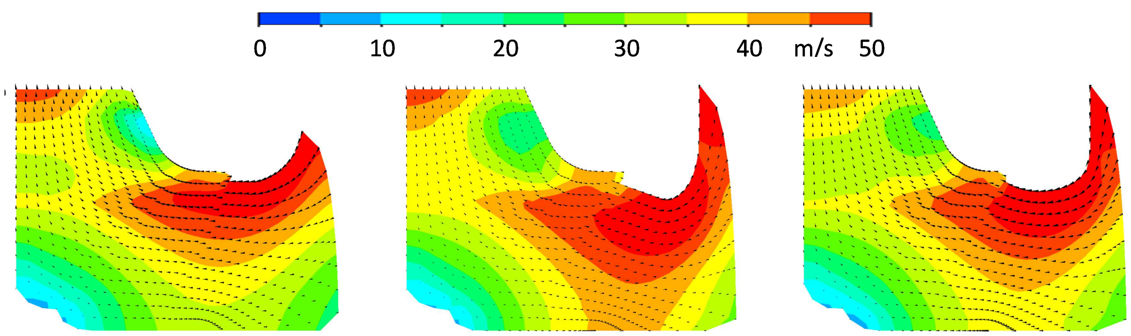
Modification of the impeller blade

The aerodynamic design of centrifugal fans allows the obtainment of the optimal inlet and outlet blade angles, but no precise guidelines are given for the choice of the blade angle variation between the inlet and the outlet section. The MP fan has a single arc of circle blade, that is the most common blade shape because it has shown good performance and is convenient from a technological point of view. Therefore in this paper the modification proposed is a "mixed" blade with single circle arc followed by a straight part (see Fig. 9). Two "mixed" blades were tested, the MP_L25 and the MP_L50, whose straight part is respectively as long as 25% and 50% of the total blade length.

From Fig. 9 it can be seen that the straight part increases the total blade length. Observing the total pressure increment comparison in Fig. 10a, the mixed blade performs slightly worse at the BEP point, but this difference increases more at a higher flow rate. For the original blade the shaft power in the simulated working range is monotone increasing, but reaching a maximum for higher flow rates is foreseen. The "mixed" blade MP_L25 shows the maximum in the simulated range and the MP_L50 shows the maximum for an even lower flow rate, so it can be asserted that the blade shape influences the position of the maximum shaft power. In general the designer can rely on the blade shape parameters to limit the shaft power variation. For example in the case of electric actuators, with constant shaft power the motor would work with constant speed and power factor without affecting the efficiency of the driving system.

Re-design of the MP blade

The redesign will be carried out with the 1D theory approach using the Stodola method for the evaluation of the slip factor of the blade. The procedure used allows the calculation, for every radius



(a) *(b)* *(c)*

Figure 8: Meridional passage circumferential averaged velocity in the rotating frame of reference, at non-dimensional flow rate Q/Q^{BEP}=1.76, respectively for the fans: a) LP, b) LP_M1 and c) LP_M2. Q^{BEP} is the Best Efficiency Point flow rate for the LP fan, calculated with CFD.

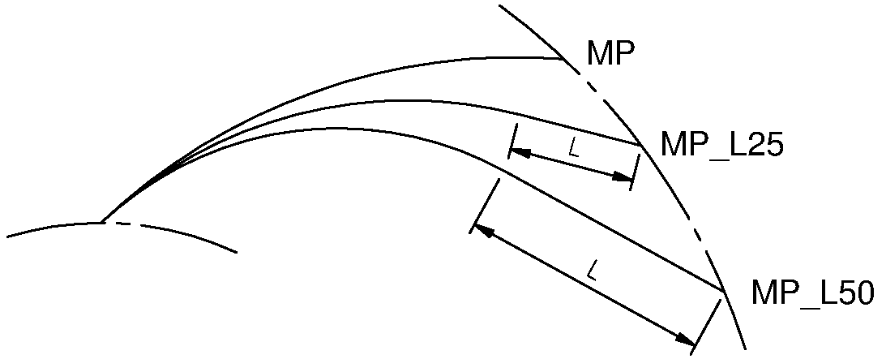


Figure 9: comparison of the MP blade with the MP_L25 and the MP_L50.

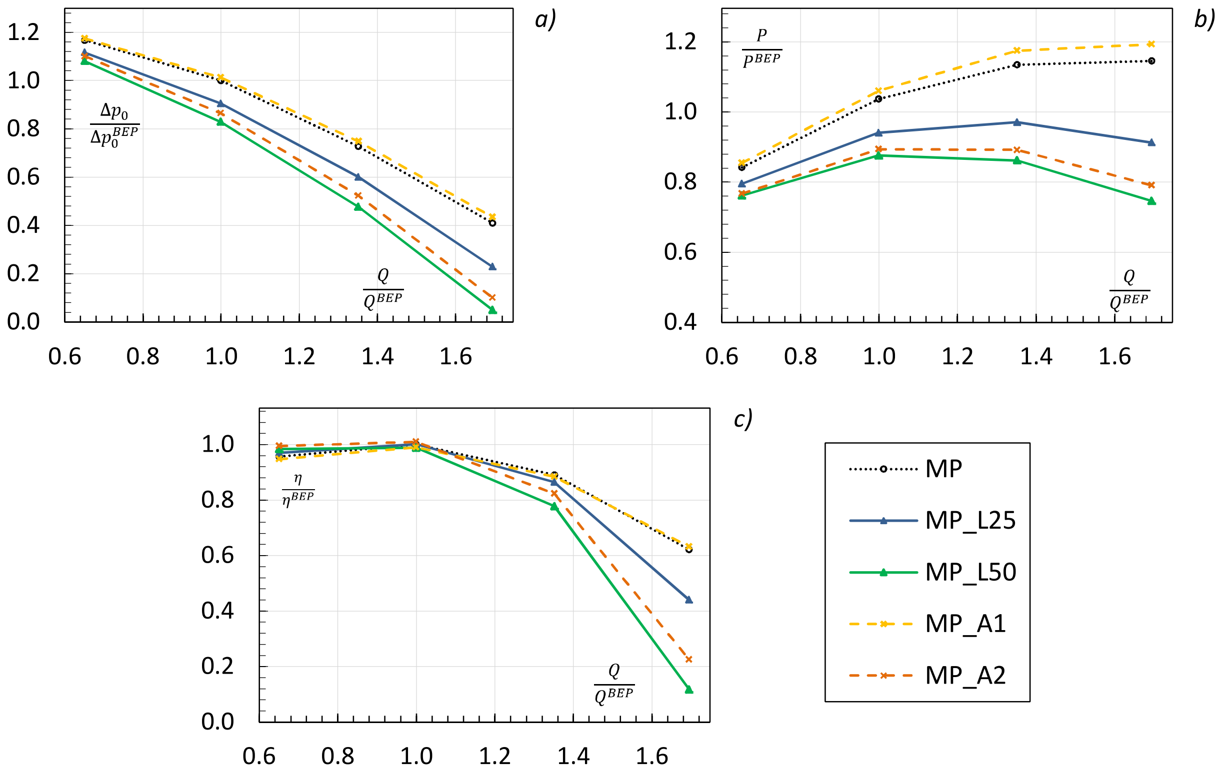


Figure 10: comparison of the non-dimensional performance of the modification and redesign of the MP fan, in terms of a) total pressure increment, b) efficiency and c) shaft power. The normalization is done with respect to the Best Efficiency Point calculated through CFD for the MP fan.

of the impeller, of the blade angle $\beta = \beta(p_0)$ that realizes the imposed total pressure increment. Two head-radius laws will be tested: a linear law for the MP_A1 fan and an exponential law for the MP_A2 fan (see Fig. 11c). The linear blade smoothly distributes the blade loading on the impeller, theoretically minimizing the possibility of flow separation (see Fig. 11a). On the other hand the exponential approach MP_A2 provides a soft load at the beginning of the passage with a rapid increase towards the exit (see Fig. 11b).

Figure 11d shows that the total pressure of the MP_A1 and MP_A2 blades varies according to the design hypothesis (linear and exponential total pressure variation), except for a slight deviation at the leading and at the trailing edge. In Fig. 11c it can be seen that the MP_A1 blade is very similar to the original MP blade and it also has a similar performance with a slight increase in efficiency for a high flow rate (see Fig 10). Fig. 10 shows that the MP_A2 has a very low performance for higher

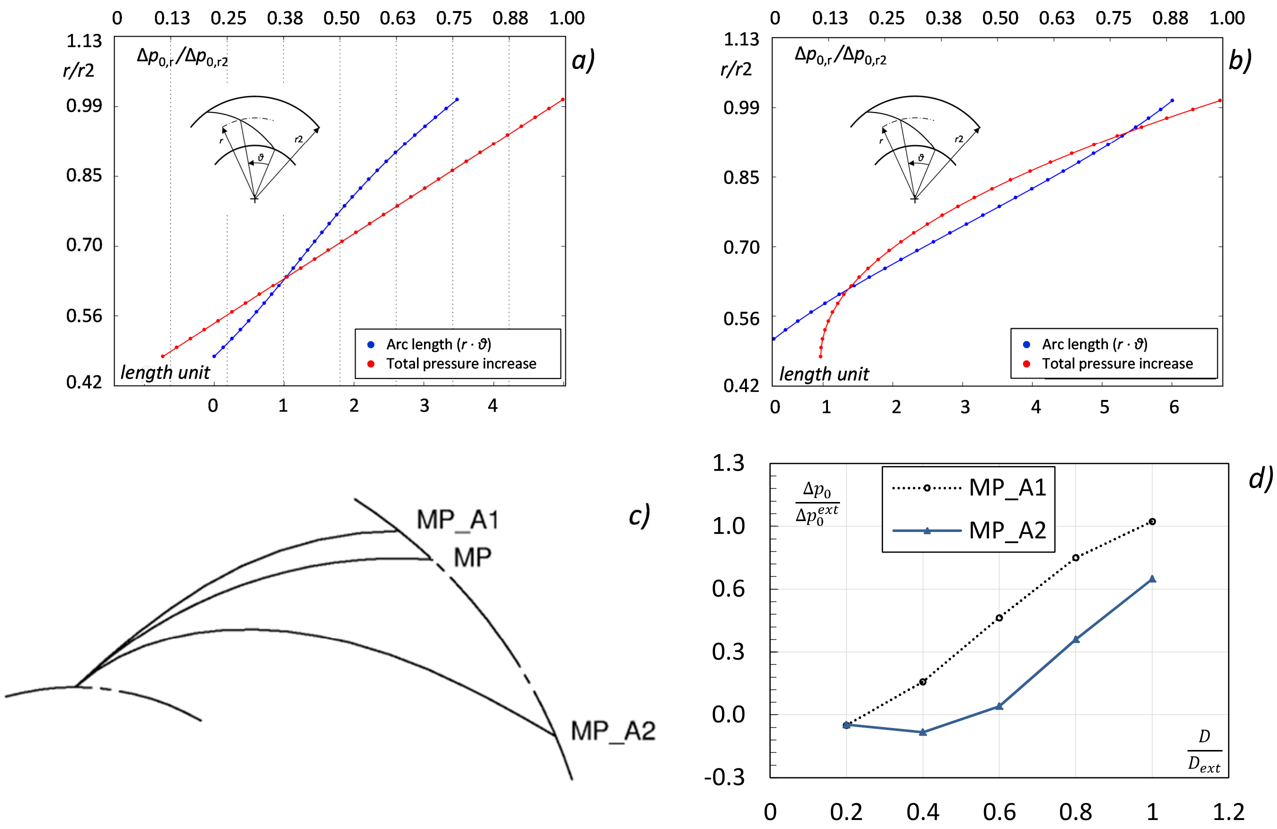


Figure 11: analytical total pressure distribution for a) the blade MP_A1 and b) the MP_A2; c) blade shape comparison of the MP blade with respect to the MP_A1 blade and the MP_A2 blade; d) comparison between the CFD total pressure distribution between the MP_A1 and MP_A2 blades.

flow rates, but a better performance for low flow rates. This suggests that the low angle variation after the leading edge reduces the separation typical for the fans that work at a flow rate lower than BEP, but also increases the blockage for high flow rates. Lastly the blade angle distribution doesn't noticeably change the design point performance, but has a big impact outside the optimal working condition.

CONCLUSIONS

The CFD was used to assist the optimization of the working range of two centrifugal fans. The meridional passage shape and the blade shape were tested to see their impact on the performance of the fans. The result shows that these parameters can alter the working range of the fans and also that CFD allows this kind of optimization. An analytical procedure for the design of the fan blade loading distribution was also suggested, and showed promising results allowing the prediction of the impeller radial loading of the blade, the modification of the fan working range and better coupling of the motor with the fan. A more thorough CFD and experimental study is needed to detect the design guidelines of the blade and meridional passage geometries which allow the obtainment of the desired working range.

REFERENCES

[1] Commission Regulation (EU) No 327/2011, **2011**

[2] Chiou, C. B., Chiou, C. H., Chu, C. M., Lin, S. L., *The application of fuzzy control on energy saving for multi-unit room air-conditioners*, Applied Thermal Engineering, **29**, pp. 310-316, **2009**.

[3] Stout, M. R. Jr., Leach, J. W., *Cooling Tower fan Control for Energy Efficiency*, Energy Engineering, **99**(1), pp. 7-31, **2002**.

[4] A. Fortini, M. Merlin, C. Soffritti, A. Suman, G.L. Garagnani, *Caratterizzazione di lamine in lega a memoria di forma NiTi per l'impiego in strutture attive deformabili*, Atti del 35° Convegno Nazionale AIM, Roma, 5-7 novembre 2014, AIM Ed., Milano, **2014**.

[5] B. Eck, *Fans*, Pergamon Press, New York, USA, **1973**

[6] A. J. Stepanoff, *Turboblowers*, John Wiley & Sons, Inc., New York, USA, **1955**

[7] Baniasadi, E., Aydin, M., Dincer, I., Naterer, G. F., *Computational Aerodynamic Study of Automotive Cooling Fan in Blocked Conditions*, Engineering Applications of Computational Fluid Mechanics, **7**(1), pp. 66-73, **2013**.

[8] Tallgren, J.A., Sarin, D.A., Sheard, A.G., *Utilization of CFD in development of centrifugal fan aerodynamics*, C631/016/2004, Proc. Int. Conf. on Fans, IMEChE Publishing, **2004**.

[9] Ferrari, C., Pinelli, M., Spina, P. R., Bolognin, P., Borghi, L., *Fluid dynamic design and optimization of two stage high performance centrifugal fan for industrial burners*, ASME Paper GT2011-46087, **2011**.

[10] Corsini, A., Delibra, G., Rispoli, F., Sheard, A. G., Venturini, P., *Aerodynamic simulation of a high-pressure centrifugal fan for process industries*, ASME Paper GT2013-94982, **2013**.

[11] Zhao, Y., Song, L., Wenqi, H., Weixiong, W., Dongtao, H., Zhichi, Z., *Numerical simulation of flow field for a whole centrifugal fan and analysis of the effects of blade inlet angle and impeller gap*, HVAC & R research, **11**(2), pp. 263-283, **2005**.

[12] Thakur, S., Lin, W., Wright, J., *Prediction of Flow in Centrifugal Blower Using Quasi-Steady Rotor–Stator Models*, Journal of Engineering Mechanics, **128**, pp. 1039-1049, **2002**.

[13] ANSYS CFX Release 14.5, User Manual

[14] James M. Sorokes, Jason A. Kopko, Paul R. Geise and Angelina L. Hinklein, *The Influence of Shroud Curvature and Other Related Factors on Impeller Performance Characteristics*, ASME Paper GT2009-60109, **2009**.