LOW WEIGHT, HIGH SPEED AUTOMOTIVE FAN DESIGN BY 3D INVERSE METHOD

M. Henner, B. Demory, Y. Beddadi, P.A. Bonnet
Valeo Thermal Systems, 8 rue Louis Lormand, 78321 La Verrière, France

F. Pengue
Advanced Design technology Dilke House, 1 Malet Street London WC1E 7JN, UK

M. Zangeneh
Department of Mechanical Engineering, University College London, London WC1E 7JE, UK.

ABSTRACT

The cooling system of modern automobiles is the subject of intense reflections to maximize efficiency and reduce the energy consumption. Large fan diameters are preferred to enhance thermal exchanges over the large surface of the radiator, whereas high rotational speeds are sought to benefit from higher efficiency and low weight of the electrical motor that drive the fan. This leads to reconsider the design of the blades for these conditions for which stagger angles are very high and the aerodynamic load very low. These unconventional geometries are so far little known but represent an interesting potential in terms of design of the cooling system, which could benefit from a high permeability favorable to the heat transfer when the vehicle is moving, even at low speed.

The aim of this paper is to apply 3D inverse design method coupled with multi-objective/multi-point automatic optimization method to design a low loaded axial fan. In this approach Design of Experiment Method (DoE) is used to create a Response Surface (RSM) relating the various performance parameters to inverse design base design parameters. Multi-objective Genetic Algorithm (MOGA) is then run on the response surface to find the tradeoffs between design parameters and constraints. All designs are evaluated by using unsteady 3D CFD simulation to create the Response Surface. An estimate of aeroacoustic far field noise is obtained by using an acoustic FW&H model. Some experimental results from a prototyped version are also presented to evaluate the accuracy of numerical simulations.

Two designs are selected by the optimization process with different compromises in term of multi-objectives performances, and their final qualities are assessed by a thermal simulation on a cooling module. These proposed new geometries represent a proof of concept that is analyzed for performance evaluation before further developments for the automotive application.

INTRODUCTION

Axial fans are commonly used to force air flow through heat exchangers, and their characteristics producing high flow rates for low pressures is well adapted to automotive radiators: the engine cooling is often given to heat exchangers with a “surface / air resistance” ratio which promotes the natural ventilation when the vehicle is cruising.

In order to increase further the thermal efficiency of the complete cooling module, it is also possible to use larger diameter fans that cover well the exchanger surface as flow rate increases with the cube of diameter. However, performances should be adapted to the nominal operating point, either by reducing the fan rpm (e.g 320 mm diameter spinning at 3000 rpm and a 460 mm at 2200 rpm), or by unloading the blade to reduce the mechanical power that would otherwise vary as power of 5 of the diameter.

Increasingly car manufacturers are trying to reduce power consumption of accessory equipment and reduce weight. For a cooling fan, most of the weight comes from the electric motor that weighs between 1 to 3 kg depending on the electric power. Increasing the fan speed of rotation is a way to improve the motor efficiency thanks to a better electromagnetic effect and lower current which help to reduce the windings and size of the permanent magnets. This means that designing automotive cooling fans with larger diameter and rotating at high speed but with low torque is a serious option for the relief of the front end of the vehicle, but this trend is accompanied by drastic changes in the fan design since the rotational speed increase should be compensated by a low loading on the blade and a low solidity of the fan. The high rotational speed and large diameter results in high tip speeds and hence relatively higher aeroacoutics noise emission. The feasibility of designing such an axial fan, therefore poses some difficult multi-objective challenges.

The aim of the present study is to conduct a 3D multi-objective optimization in order to design a lightly loaded axial fan operating at high speed and with a good compromise between performance at both nominal point and high flow rate, acoustic emissions. A 3D inverse design method that computes the blade geometry according to aerodynamics parameters is used in combination with numerical simulations for different operating points and with an assessment of the sound pressure level. A design of experiment is conducted with 7 parameters describing the aerodynamic loading, and surface responses are created to search optimum solutions with the help of a genetic algorithm.

In order to push the concept to its extreme, a rotational speed of 3600 rpm is assumed for a fan having a diameter of 420 mm. The tip velocity of almost 80 m/s is of course a concern for the acoustic and some precautions would have to be taken to validate this high rotational speed. Meanwhile the proposed fan design would help to evaluate the concept, in particular for automotive applications where the thermal performance of the cooling module is one of the validation criteria. Finally, some thermal predictions of performances are presented to demonstrate the cooling efficiency.

STATE OF THE ART

Such low loaded fans are quite unconventional and little is known about them. Many studies have been carried out for classical axial fans that can be designed by quasi 3D methods, such as the one described in [1] and based on some *a priori* assumptions to describe the meridional flow with concentric layers. For axial turbomachinery, different models of spanwise vortex distribution can be used to obtain the condition of radial equilibrium at the exit of the fan. However, such a condition could be difficult to meet for lightly loaded fans. A typical consequence of that would lead to an inability to operate correctly at the hub. Other methods dealing with axial machines can propose modified vortex models that can improve the solution as proposed by [2], and which has been the subject of experimental studies by [3] and [4]. These methods are advantageously supplemented by numerical simulation, allowing precisely diagnosing the possible causes of losses such as separation and/or recirculation. These effects have been studied for both acoustics and aerodynamics purposes by [5] and [6], who have shown some separations on the suction side of the blade near the hub and who propose to improve the situation either by changing the shape of the hub or by using different blade curvatures.

Experimental and numerical studies on cooling modules have confirmed the difficulty to maintain an "ideal" flow without recirculation, especially when some disturbing obstacles such struts or arms are present close to the fan. Measurements made by [7] demonstrated the existence of a quite high amount of flow recirculation which can be explained by the presence of peripheral clearance and the difficulty to maintain enough total pressure at blade hub. It can lead to a high level of separation on the upper side, or at least to a thickening of the boundary layer close to the trailing edge. In addition to lower efficiency, it might increase the fan self-noise as it has been shown by the model of [8] which assesses the self-noise by the boundary layer thickness at trailing edge.

NUMERICAL DOE ON INVERSE DESIGN METHOD AND OPTIMIZATION

Low loaded fans are barely known and there is little previous experience on the design of this type of fan. A classical specification for fan system has been selected, with the following targets:

- An operating point at 2500 m3/h requiring a static pressure rise of 220 Pa (mandatory)
- An off-design point at 4000 m3/h requiring a maximisation of the pressure
- An minimisation of the torque (or a maximisation of the static efficiency)
- An minimisation of the sound pressure level

Inverse design method

Initially an attempt was made to design a baseline fan rotor for these conditions by using a conventional design approach. However, this resulted in a fan geometry with rather poor efficiency. Hence, the 3D inverse design method, TURBOdesign1[9] was used to design a baseline rotor. This code was also used to parametrically describe the blade geometry for automatic optimization. TURBOdesign1, is a three-dimensional inviscid inverse design method, where the distribution of the circumferentially averaged swirl velocity (rV_θ) is prescribed on the meridional channel of the blade and the corresponding blade shape is computed iteratively.
The circulation distribution is specified by imposing the spanwise (rV_θ) distribution at leading and trailing edge and the meridional derivative of the circulation $d(rV_\theta)/dm$ (blade loading) inside the blade channel. The input design parameters required by the program are the following:

- Meridional channel shape in terms of hub, shroud, leading and trailing edge contours.
- Normal/tangential thickness distribution. The thickness distribution is fixed during the inverse design process.
- Fluid properties and design specifications.
- Inlet flow conditions in terms of spanwise distributions of total temperature and velocity components.
- Exit (rV_θ) spanwise distribution. By controlling its value, the work coefficient (rotor) or flow turning (stator) are controlled (see Fig. 1). Totally arbitrary spanwise circulation distributions can be specified by using b-splines.
- Blade loading distribution $d(rV_\theta)/dm$. It is imposed at two or more span locations. The code then automatically interpolates spanwise to obtain the two-dimensional distribution over the meridional channel. In figure 2, a typical loading distribution at the hub and tip are presented. The blade loading is specified by using a 3-segment approach in which a parabolic loading is used from leading edge to a point NC, followed by a straight line section, the slope of which can be specified by the user. This is then followed by another parabolic section that brings the loading to zero at trailing edge. At the leading edge, for no

incidence zero value of loading or DRVT can be specified. However, it is also possible to specify a positive incidence or negative incidence by using non-zero values of DRVT at the leading edge. In this way the loading on each streamline can be controlled by maximum of 4 parameters, NC and ND for start and end point of the parabolic section and Slope for the slope of the straight line section and DRVT for incidence on the leading edge. In most cases, the location of NC and ND can be kept fixed and DRVT and Slope are the only parameters that are modified.

- Stacking condition. The stacking condition must be imposed at a chord-wise location between leading and trailing edge. Everywhere else the blade is free to adjust itself according to the loading specifications. The stacking condition can affect the blade sweep and can have a significant effect on the 3D pressure field in the fan.

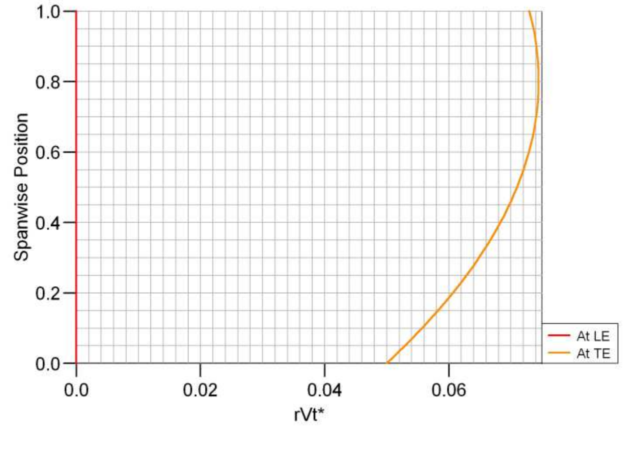

Figure 1 : R^*V_t distribution

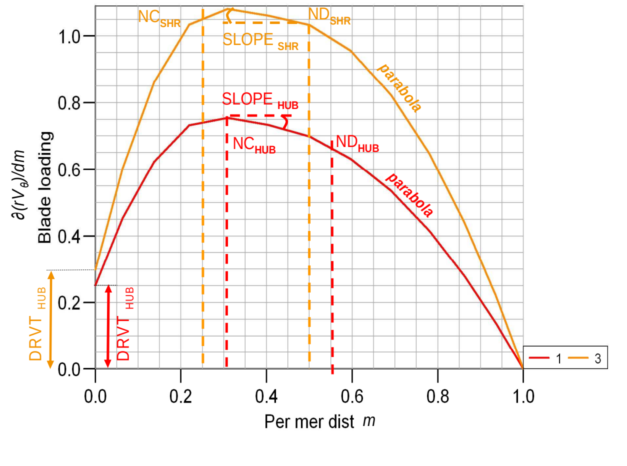

Figure 2 : Profile loading description with 4 parameters

DOE with aerodynamic parameters

Some optimisation processes based on the direct parameterization of the fan blade geometry have already been presented by [12] and [13], and the technique that is presented in this paper differs by the method used to parameterize the blade geometry, which is based on inverse design code. The inverse design based parameterization uses blade loading rather than the blade shape. The approach can generally cover larger design space with much fewer design parameters as compared to direct blade geometry parameterization, see [9]. Furthermore, in Design of Experiment (DoE) based optimization this approach can lead to more accurate response surfaces (or surrogate models) which can then be used for efficient multi-objective optimization, see [9] and [15].

Design Parameters

The fan geometry was parameterized by using the blade loading and stacking parameters. For blade loading the values of NC at the hub and shroud were kept fixed but the values of ND, Slope and DRVT at leading edge at the hub and shroud were varied. In all optimization problems the correct range of variation for design parameters is an important choice for setting up the problem. By using inverse design based optimization the correct choice of parameters can be visually explored with ease. In this case the range was varied as shown in Table 1.

Table 1: Range of variation of Loading parameters

ND_{HUB} & ND_{SHR}	0.3 to 0.9
$SLOPE_{SHR}$ & $SLOPE_{HUB}$	-10 to 10
$DRVT_{SHR}$ & $DRVT_{HUB}$	0 to 0.9

In addition to the 6 parameters used to control the blade loading, one parameter was used for stacking. The stacking location was fixed at trailing edge and the value of wrap angle at the hub was varied from -12 to +12 degrees while the value at the shroud was kept fixed at 0 degree. 12 degrees actually represents about 55% of total wrap angle change between leading and trailing edges for the designed fan. So in total 7 design parameters were used to represent the 3D blade geometry of the fan.

Performance Parameters

The choice of key performance parameters were taken at nominal design flow rates and high flow rates as summarized in table 2.

Table 2: Key Duty points for baseline Rotor

	Flow Rate (m^3/hr)	Noise	Efficiency	Pressure Rise (Pa)
Duty Pt 1	2500	Minimize	Maximize	> 200 Pa
Duty Pt 2	4000	-	-	Maximize

The design matrix was generated by using Orthogonal Latin Hypercube (OLH) plan with 36 configurations. The resulting geometries were created by using TURBOdesign1 and then meshed for CFD computations.

Numerical simulations

Numerical simulations are conducted on the various designs with the commercial CFD code CCM+ from CD-Adapco. Polyhedral meshes are used in the domain of simulation which includes a rotating domain around the fan (Figure 3), and cell extrusions are used on all walls to resolve the boundary layers. The number of polyheadra is about 3 millions for the complete domain, and the finer area is located in the rotating domain which concentrates 2 millions nodes in this region, see Figure 4. The Y+ value is close to 1 on the fan and on the shroud at rest, allowing the use of a two-layer model for boundaries layers. Extrusions are coarser on walls that are far from the fan. The two equation turbulence model k-ω SST from Menter is used to predict dissipation and length-scale for turbulence.

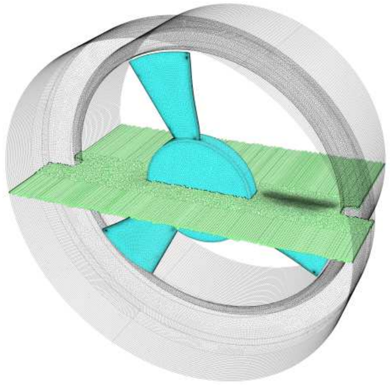	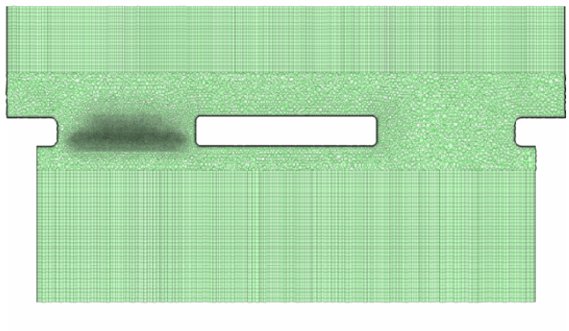
Figure 3: in-duct domain of simulation	Figure 4 : cross-section of the mesh

In order to obtain a measure of far field noise from the fan, unsteady RANS simulations are conducted which are then used together with the FWH [10] model already implemented in CCM+ with rigid body motion (rotating sliding mesh). A full 360 degrees mesh is considered and a steady-state RANS solution is used to initialize the transient simulation. The time step during the simulation is 4.63e-04 seconds. The CPU cost of every simulation is ~500 hours CPU time. The dipolar sources are extracted from unsteady forces that occur on moving walls and a spectrum of acoustic pressure is build at a given position of an observer located on the axis of rotation, in front of the fan at a distance of 1 meter. Unsteady results are used for aeroacoustic sources after 5 revolutions in order to avoid the initial numerical transients. A global noise level is assessed from frequencies below 750 Hz, which correspond to a reasonable cut-off frequency regarding the discretization in time and space. It indicates levels accounting for the 3 first harmonics with the formula $N = 20 * \log_{10}(\sum_i 10^{(\frac{Spl_i}{20})})$, the blade passage frequency being 180 Hz. This acoustic level should be seen as the bare minimum noise from dipolar sources that could be produced by such a fan since the simulation does not take into account non-homogeneity of the inlet flow and the only effect that increases the acoustic pressure is the tip clearance recirculation.
The global performance of each geometry is extracted in terms of pressure rise at nominal and high flow rate and in terms of torque and efficiency. Efficiencies are computed from the static pressure and/or the total pressure.

Response Surface and Optimization

All the results have been summarized in a table and used to build a surface response from the 36 runs. A quadratic response surface was set up, neglecting some of the less important interaction terms. The resulting least square regression seems to fit the data with high degree of accuracy as shown in Figure 5 and 6 for pressure rise, torque and far field noise at the design point. Both the squared residuals analysis (difference between actual and predicted value) and the prediction analysis are presented in figure 5 and 6, respectively.

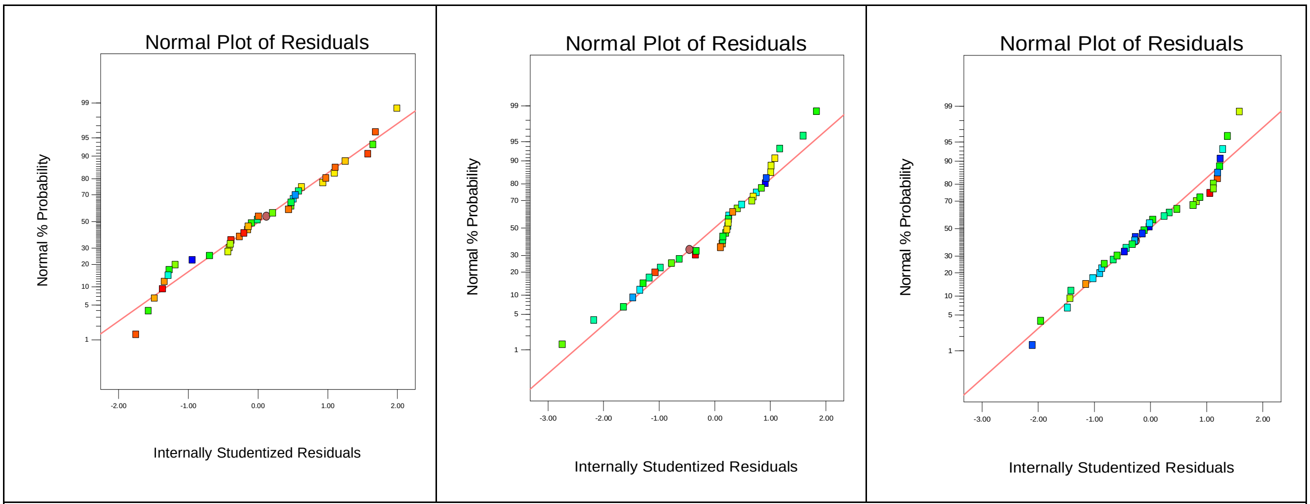

Figure 5: squared residuals analysis for pressure rise (left), torque (middle) and acoustic criteria (right)

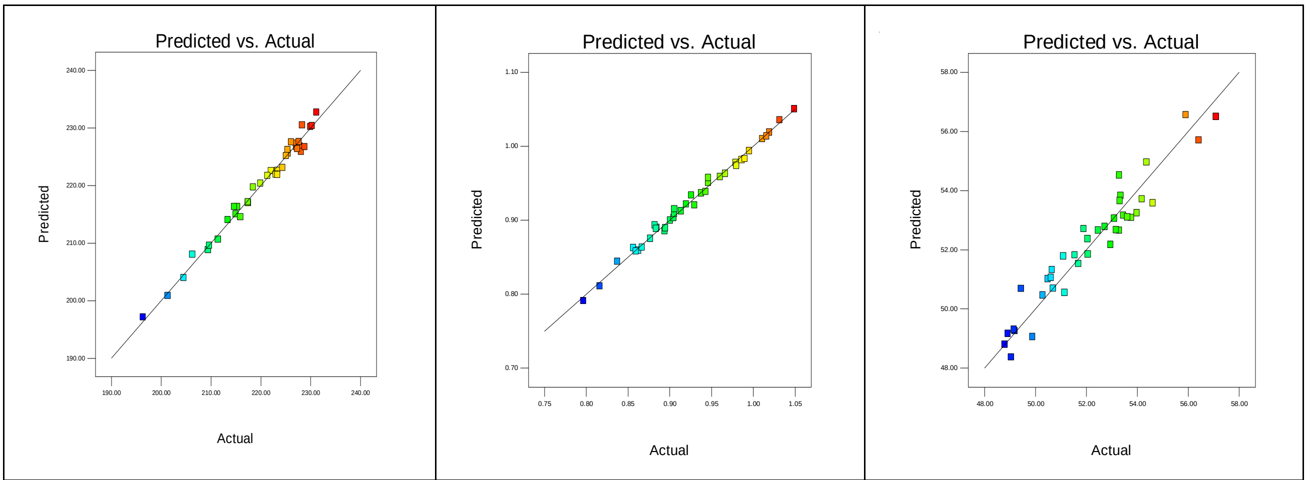

Figure 6: prediction analysis for pressure rise (left), torque (middle) and acoustic criteria (right)

Residuals for the 3 response surfaces show that some dispersion exists between the model and the values obtained by simulation. However the comparison between the prediction and the actual values shows that the model capture pretty well the physics and at least the trend can be considered reliable. It is less obvious for the acoustic criteria based on the FW&H analogy, and it must be admitted that this surface response is probably less accurate.

The final step consisted of using the Genetic Algorithm in Isight [14] on the Response Surface to find the best solution. Non-sorting Genetic algorithm type 2 was used for optimization. The response surface was used to evaluate objective functions and the whole optimization process could be completed in a few seconds. The criteria for optimization combines multi-objective criteria based on best efficiency at nominal point, high pressure rise at high flow rate and low sound pressure level. At the same time a minimum pressure rise of 220 Pa was specified as a constraint at nominal point. Two different designs are proposed at the end for further analysis, and their performances are summarized in Table 3.

Table 3: Key performances of the Design1 and 2 selected from Optimization

	Static efficiency at nominal point (%)	Pressure rise at nominal point (Pa)	Acoustic criteria based on FW&H analogy	Pressure rise at high flow rate (Pa)
Design1 RSM Values	*45*	*225*	*52.4*	*93*
Design 1 CFD Validation	43	224	51.7	98
Design2 RSM values	*46*	*229*	*53.9*	*88*
Design 2 CFD Validation	48	229	52.1	47

In table 3 basic performance parameters of the two fan geometries that were selected from the optimization for further investigation are presented. These designs are labelled as Design1 and Design2. For validation all steady and unsteady CFD computations were repeated for these two designs. The comparison between the RSM and CFD results presents a good correlation in terms of pressure rise at nominal point and the acoustic criteria based on FW&H analogy. However, for Design2 there is a discrepancy in the pressure rise at high flow rate. Further analysis indicates that this point is actually close to the boundary of the DoE table for this performance parameter and hence some of the value may have been extrapolated in the response surface.

Each of them has been selected for their strength, i.e. for the first design the pressure at high flow rate, and for the second one its higher efficiency. For information, pictures of each of them are presented in figures 7 and 8, respectively.

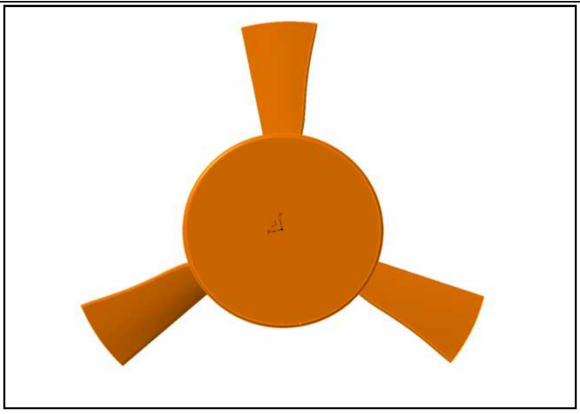

Figure 7: Front view of Design 1

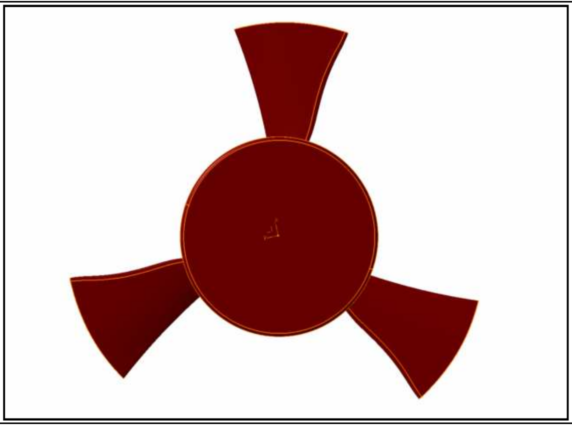

Figure 8: Front view of Design 2

VALIDATION AND COOLING MODULE SIMULATION

Fan alone on test rig

Further analyses have been conducted to check actual performances of the fan with the help of numerical simulation with the same software STAR-CCM+ (CD-Adapco). Same conditions have been used for the numerical set-up, except that the domain of simulation is extended to get a proper consideration of the far field, both from the upstream and downstream side. This domain is presented in Figure 9, and the results for global performances are shown in figure 10: CFD predictions of performance for Design1 and Design2 are presented and compared with Design3. Design3 was actually a baseline design generated by using the inverse design code TURBOdesign1. This geometry, which has the same chord but smaller hub radius, was machined from aluminium and then its performance was measured in the test facility. The comparison of CFD results with measurements for Design3 show that the CFD predicts the fan performance quite well and hence we can rely on the predicted performance for Design1 and Design2. Same comparisons for torque presented in Figure 11 show bigger discrepancies with an under estimation of the torque of about 15 to 20% at low flow rate, which of course leads to an over prediction of the efficiency on Figure 12. However some points have to be emphasized:

- The bell curves of efficiency for both Design 1 and 2 show a wider range of efficiency compared to the Design 3 obtained directly by the inversed design method
- Design 1 provides more pressure rise and this explains the higher performance at high flow rate. Of course, torque requirement for the fan is also bigger
- Design 2 provides a slightly higher efficiency at the nominal operating point (2500 m3/h) but provides less performance for the high flow rate. These results indicate that at least the ranking from the optimization process is correct and confirmed by the test rig simulation

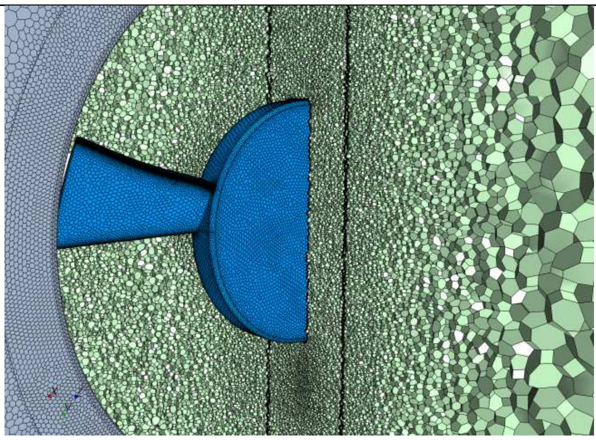

Figure 9: extended domain of simulation to reproduce the test-rig conditions

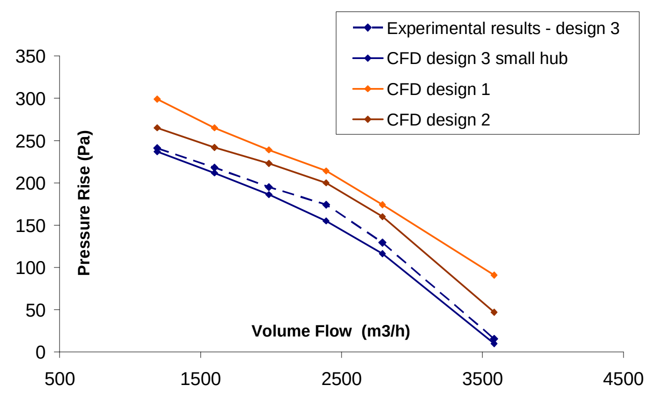

Figure 10: comparison of numerical and experimental results for pressure rise

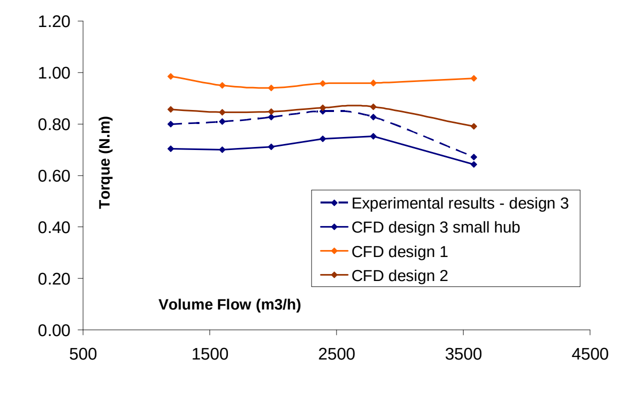

Figure 11: comparison of numerical and experimental results for torque

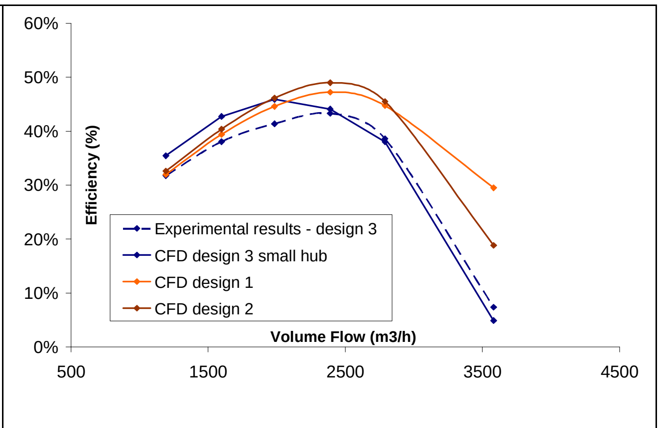

Figure 12: comparison of numerical and experimental results for efficiency

Thermal simulation for cooling module

An assessment of actual thermal performance for the two optimized fans has been performed with numerical simulation that includes a heat exchanger with a transverse flow of coolant (Figure 13). The feature "dual-stream" of CCM + is used to evaluate the thermal performance, and an inlet flow of liquid at 95 ° C is used in the heat exchanger, this latter being modeled for the aerodynamic side by a porous media. The fan is positioned behind the heat exchanger and the simulation domain is designed to represent an environment of automotive underhood: it comprises an upstream air inlet with a bumper and a downstream obstacle representing the engine of the vehicle. The volume is meshed with polyhedra and simulated using the same CCM+ methodology already presented. To have a fair comparison, identical inlet total pressures are imposed as a boundary condition and both the flow rate and the thermal performances are given by the simulation.
The figure 14 shows the heat exchanger for the Design 1, where the effect of the air entrance shape and the different obstacles can be seen with the heterogeneous temperature distribution on its inlet face.

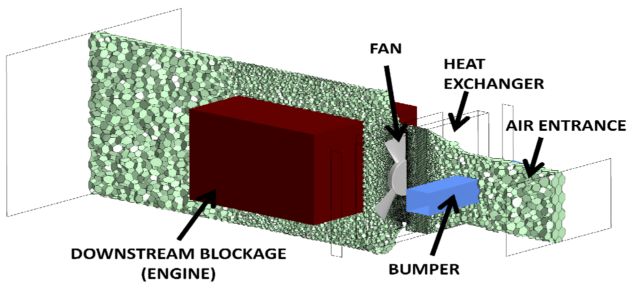

Figure 13 : cooling module simulation with upstream and downstream obstacles

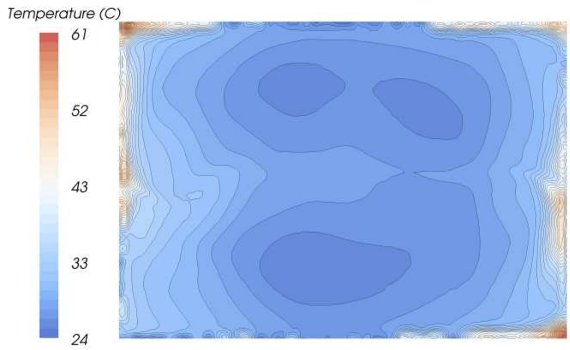

Figure 14 : temperature distribution on the heat exchanger for Design 1

Global results are shown in term of heat rejection (kW) and mechanical power for the same flow condition:

	Flow rate (m3/h)	Heat rejection (W)	Mechanical power (W)	Ratio Heat/mechanical power
Design 1	2450	23540	274	86
	3416	26900	281	96
Design 2	2410	23400	238	98
	3380	26500	246	108

It can be seen that the final thermal performances are rather identical, and that the Design 1 does not outperform so much Design 2 despite its initial higher pressure rise. This effect could be explain at least at nominal flow rate by the higher efficiency of the second design which might compensate performances with a better integration in the cooling module. It is further demonstrated by the ratio of the heat rejected by the mechanical power which indicates the amount of energy to be subtracted on the powertrain for the same cooling and which is again in favor of the second design.

CONCLUSIONS AND PERSPECTIVES

An unconventional automotive axial fan design with low torque and high rpm, which can result in reduction fan motor weight, was rapidly developed by using a 3D inverse design method with significantly higher efficiency that could achieved by using conventional design approach. This design was further improved by coupling the 3D inverse design method with an automatic optimization strategy based on Design of Experiments, Response Surface Modeling and then running a Genetic Algorithm on Response Surface. The method was used to optimize efficiency and noise at design point and pressure rise at high flow rate. Using this approach two designs were selected for further investigations. One design had higher pressure rise at high flow rate while the other had higher efficiency at the design flow. Their global performances have been finally been checked by a thermal simulation in a cooling module with geometry of air entrance and downstream domain that mimics underhood conditions. Thermal prediction results show that the amount of heat rejected by the module is equivalent to the values obtained with a classical fan. Further studies can then be considered to check the relevancy of the solution in a car. Acoustic data collected experimentally will help assess the limitation of this concept regarding noise.

REFERENCES

[1] « Turbomachines : calcul des écoulements compressible », Meauzé G ; Techniques de l'Ingénieur (1995) vol. BL2 num. B4181

[2] « Ventilatoren: Aerodynamischer Entwurf, Schallvorhersage, Konstruktion", Carolus T., Vieweg+ Teubner edition

[3] "Experimental and numerical study of the sweep effect on three-dimensional flow downstream of axial flow fans", Hurault J., Kouidri S., Bakir F., Reya R. Bakir F." Flow Measurement and Instrumentation 21 (2010) p155-165

[4] Vad, J., Bencze, F. (1998), "Three-Dimensional Flow in Axial Flow Fans of Non-Free Vortex Design", International Journal of Heat and Fluid Flow, Vol. 19, pp. 601-607.

[5] "Optimization of Axial Fans with highly swept blades with respect to losses and noise reduction", Bamberger K., Carolus T., FAN 2012 Senlis (France), 18 – 20 April 2012

[6] "Aeroacoustical Investigations on Axial Fans for Automotive Cooling Systems", Zayani M., Çaglar S., Gabi M., FAN 2012 Senlis (France), 18 – 20 April 2012

[7] "Comparison of experimental and numerical flow field in an automotive engine cooling module", Henner M., Moreau S., Brouckaert J.F., European Turbomachinery Conference ETC 8, 23-27 march 2009, Graz Austria

[8] "Rotating blade trailing-edge noise: experimental validation of analytical model", Rozenberg Y., Roger M., Moreau S., (2010), AIAA J. 48, 951–962

[9] "On the Coupling of Inverse Design and Optimization Techniques for the Multiobjective, Multipoint, Design of Turbomachinery Blades", Bonaiuti D., Zangeneh M., Transactions of the ASME, J. of Turbomachinery, 021014-16 / Vol. 131, APRIL 2009.

[10] "Sound generation by turbulence and surfaces in arbitrary motion, Philosophical Transactions for the Royal Society of London", Ffowcs Williams, J. & Hawkings, D. (1969). A264: 321–342.

[11] "Acoustic radiation from an airfoil in a turbulent stream", Amiet R. K., Journal of Sound and Vibration, 41(4):407–420, 1975.

[12] "Influence of different parameters on Pareto fronts used for the optimization of a fan blade by coupling a genetic algorithm and a flow parameterization software Turb'Opti", Buisson M., Ferrand P., Soulat L. Rambeau C. Aubert S., Henner M., European Conférence on Computational Mechanics 2010

[13] "Design and off-design optimization of a fan for automotive applications", T. Verstraete T., Roytta P., Henner M., Alsalihi Z., Brouckaert J.F., European Turbomachinery Conference 2011

[14] Isight / SEE 5.7 User's manuel.

[15] "Optimization Of Fan Noise By Coupling 3d Inverse Design And Automatic Optimizer", Zangeneh, M and Maillard, M., FAN 2012 Senlis (France), 18 – 20 April 2012