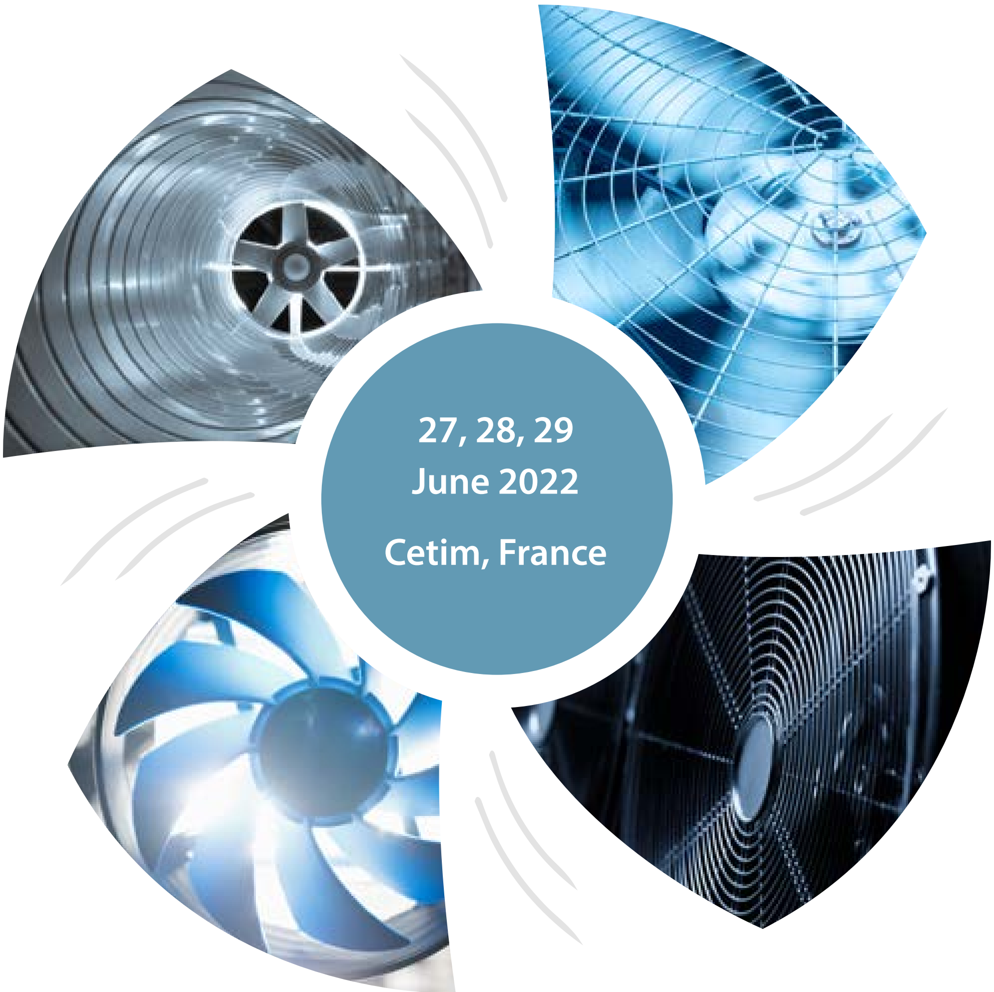
fan2022

International Conference on Fan Noise, Aerodynamics, Applications and Systems

#fancongress2022



www.fan2022.org

ABSTRACTS BOOK

CETIAT
CARNOT
cetim

fan2022

Investing in people and technology

For more than 100 years Ziehl-Abegg has had a reputation for producing durable and efficient fans. Development based on sound research and a high degree of vertical integration are characteristic of the way the company operates. The superb technical facilities at the company's locations around the world provide the ideal workplace for bright minds. One example of this is the world's largest combined measuring and test rig for fans which Ziehl-Abegg operates at its headquarters in Southern Germany.

Since the change in demographics mean that the human factor will increasingly become the limiting factor, the company is breaking new ground in attracting young talent: students are enjoying the thrill of the ZA-Games, Ziehl-Abegg's presence on TikTok is being liked by millions of users and gamers worldwide are taking part in the Royal eSports Masters.

"Having motivated and highly-trained employees allows us to look to the future with confidence" says Chief Development Officer Dr. Sascha Klett happily. Because it's only those who continually innovate that will have the right products to meet their customers' requirements in the future.

For further information go to www.ziehl-abegg.com

eSports is key to inspiring the next gen

Introduction from the conference chair

Welcome to Fan 2022. The Fan 2022 conference offers you a unique opportunity to network with those working in the air movement community. This three-day conference includes keynote lectures and technical presentations organized as three parallel sessions focused on fan application and systems, fan noise and fan aerodynamics.
The conference exhibition provides you with access to those organizations working for and with the fan industry. I would encourage you to speak with the exhibitors about your business and the challenges it faces. Their business is helping you solve your business problems, and that is why they have come to Fan 2022.
The location of Fan 2022 conference, Senlis is a French commune in the province of l'Oise near Paris. It has a long and rich heritage. The medieval town is where the monarchs of the early French dynasties lived, attracted by the proximity of the Chantilly Forest. I am sure that you will enjoy not only the conference itself, but also the location.
This is a particularly important time for the air movement community. Fan efficiency is regulated in the process of being revised in Europe. In the US regulatory effort look as if they will finally come into force in 2023. Within Asia there is also momentum towards fan efficiency regulation, but locally developed, with regulations based on FMEG, FEG, or an alternative metric. For those fan companies operating in more than one region, there is the very real prospect of having to comply with multiple fan efficiency regulations. By attending Fan 2022 you take an important step towards embracing the regulatory challenges our community faces. I look forward to welcoming you to Fan 2022.
I would also like to take this opportunity for and on behalf of Fan 2022 organizing committee to welcome you to Senlis, and to thank you for attending Fan 2022. Take your time to attend the sessions, listen to what is said and challenge what you hear. Above all, renew old acquaintances, form new friendships and enjoy the conference.

Geoff Sheard
Chairman – Fan 2022 Organizing Committee

Content

Keynote Lectures

A1 Materials & Wear Resistance

Chairmen: Peter Pelz, Johannes Brötz

A2 Influence of Tip Gap or Axial Gap

Chairmen: Henrik Witt, Alain Guédel

C3 Fan Installation & Selection

Chairman: Alain Guédel

D1 Fan Design

Chairmen: Claus Haendel, Dario Brivio

D2 Serrations, porous material

Chairman: Michel Roger

D3.1 Regulations

Chairmen: Geoff Sheard

D3.2 Fan Testing

Chairmen: Geoff Sheard, Ossian Kaghad

E1 Methods and Instruments for Transient Phenomena

Chairman: Damian Vogt

fan2022

F1 Aerodynamic Design

Chairman: Martin Böhle

F2 Sound Quality

Chairmen: Saïd Naji, Michel Besombes

G3.1 Inlet Turbulence

Chairman: Dario Brivio

G3.2 Ceiling & Circulating Fans

Chairman: Geoff Sheard

Abstracts

NOISE AND SOUND QUALITY OF FANS: HOW TO ANALYZE WITH CONSIDERATION OF THE NEW SOUNDSCAPE STANDARD ISO 12913

Klaus GENUIT HEAD acoustics GmbH, Herzogenrath (Germany)

Fan noise is everywhere, inside and outside of buildings. With respect to the increasing applications of climate control we expect more and more installations of fans. Fan sound could be disturbing on the on side but could be used as masking sound of other disturbing sound sources on the other side. Soundscape describes the acoustic environment as perceived or experienced and/or understood by people, in context (ISO 12913-1). The specification ISO/TS 12913-2 provides information about data collection and reporting requirements regarding soundscape studies, investigations and applications, and ISO/TS 12913-3 provides guidance on how to analyze the collected data. This is the first standard requesting normative the use of binaural recording and loudness analysis in combination with further recommended psycho-acoustical parameters including interviews of involved people. This paper gives an overview of the application of this Soundscape standard for the evaluation and assessment of fan noise.

ISO 12913-1:2014 : Acoustics — Soundscape — Part 1: Definition and conceptual framework

ISO/TS 12913-2:2018 : Acoustics — Soundscape — Part 2: Data collection and reporting requirements

ISO/TS 12913-3:2019 : Acoustics — Soundscape — Part 3: Data analysis

Klaus Genuit received his PhD in 1984. He developed a new, improved binaural measurement system for the advanced diagnosis and analysis of sound. He founded the company HEAD acoustics GmbH 1986 which is today a leading contributor in areas of binaural signal processing, sound design and analysis, virtual reality, and telecommunication measurements. Klaus Genuit has published more than 300 scientific papers. He is a member of various associations, such as AES, JAES, JSAE, SAE, DEGA and ASA where he was elected as a fellow in 2004. He is distinguished member of I-INCE and participates in several working groups dealing with the standardization of measurement regulations and psychoacoustic parameters. He has worked within different EU-supported research all of them focusing on improving sound quality of vehicle exterior noise. More than 15 years ago he started his investigations with respect to soundscape. Klaus Genuit is honorary professor at the RWTH Aachen University and has established the HEAD-Genuit-Foundation at 2008. He is president of HEAD acoustics GmbH and HEAD-Genuit-Foundation.

Klaus.Genuit@head-acoustics.com

WIND TURBINE NOISE AND STATE-OF-THE-ART OF NOISE REDUCTION TECHNOLOGY

Franck BERTAGNOLIO DTU, Wind Energy Airfoil and Rotor Design, Kgs. Lyngby (Denmark)

Wind energy should contribute significantly to the energy transition toward a decarbonized society in the next few decades. Although plans are being devised to deploy large offshore wind farms, onshore wind turbines are still very competitive in term of installation and operational costs. The density of onshore wind farms will therefore certainly increase in some areas. Noise regulations can result in high curtailment rates and energy yield losses. Noise plays also an important role for social acceptance of wind energy.

The lecture will start with general aspects of wind energy, such as technology, future trends and challenges, in particular related to noise. The focus will move toward the specificities of wind turbine generated noise. Mitigation techniques currently used in the industry will be addressed. Finally, latest developments and cutting-edge technology for noise reduction will be described, with a future outlook on possible advancements in this domain.

Franck Bertagnolio has been working since 1997 in the field of wind energy as a researcher at the former Risø National Laboratory, nowadays DTU (Technical University of Denmark). His initial focus was on Computational Fluid Dynamics for investigating the aerodynamic and aeroelastic properties of wind turbine blades. Progressively, his scientific interest has moved toward the noise emission of wind turbines in parallel with the rapid development of the wind power industry, and because of the close connection between aerodynamics and aeroacoustics. His main research has been directed toward the development of engineering models for wind turbine aerodynamic noise, as well as acoustic measurements in the field.

frba@dtu.dk

ENDLESS FIBRE REINFORCED COMPOSITE-METAL-IMPELLER: HIGH SPEED BURST TESTING -DAMAGE AND FAILURE ANALYSIS

Sebastian SPITZER	Technische Universität Dresden (Germany)
Richard GROTHE	Technische Universität Dresden (Germany)
Maik GUDE	Technische Universität Dresden (Germany)
Martin POHL	Technische Universität Dresden (Germany)
Peter HERMERATH	Piller Blowers & Compressors GmbH (Germany)

The performance and efficiency of future and existing blowers can be increased by using new types of high-performance impellers made of fibre reinforced materials. Such composite materials have outstanding density related mechanical properties, which allow the speed to be increased significantly. As a result of the reduced mass such impellers, in particular the reduced inertia, the impellers can react faster to variable speeds and consume less energy. Composites allow a variable design of the blades. This allows flow-optimised cross-sections to be realised, which in addition to higher efficiency also has the potential to reduce noise. Furthermore, the layered design of such impellers can be used to integrate sensors as a basis for component monitoring and data generation.

Due to the special characteristics of fibre composite materials with their large number of adjustable parameters and the special knowledge required for this in combination with high expenditures and risks for the development and production of integral rotors, these are currently only niche products.

At the TU Dresden, in cooperation with the Forschungsvereinigung für Luft- und Trocknungstechnik (FLT), a modular design of a high-performance impeller was developed, for which the functionality was presented in a first FAN paper. Compared to the integral solutions, the manufacturing effort of this modular design is greatly reduced, which allows a higher availability and variability in production. In addition to the design, a numerical-based methodology for mechanical design was elaborated and presented. This allows the consideration of material-specific damage and failure behaviour. Based on this virtual component development, speeds of about 7,500 rpm were already achieved in the first test, which corresponds to a circumferential speed of about 400 m/s and thus corresponds to a speed increase of about 30% compared to the metallic variant.

In this paper, results of the spin test up to failure at up to 538 m/s are presented. Based on high speed images taken during the spin test and fracture analysis, it is possible to interpret the damage and failure behaviour of the structure and compare the results of the simulation. A focus of the discussion is on the selected joining technique in relation to the bursting speed and an analysis of varying bursting speeds. This results in measures and indications for the further improvement of the design and for the further increase of the performance and reliability of such structures.

The smaller and lighter fragments resulting from the bursting are then considered in the discussion of the safety-related advantages of the overall system. The results of this simplified feasibility study show a clear potential for increasing the speed of high-performance radial fans in metal-fibre composite design compared to metal impellers.

Keywords: composite material, radial fan, hybrid design, high speed burst test, failure analysis

sebastian.spitzer@tu-dresden.de

ENDLESS FIBRE-REINFORCED COMPOSITE-METAL-IMPELLER: INVESTIGATION AND COMPARISON OF THE DAMPING BEHAVIOUR

Martin POHL Technische Universität Dresden (Germany)
Sebastian SPITZER Technische Universität Dresden (Germany)
Martin DANNEMANN Westsächsische Hochschule Zwickau (Germany)
Peter HERMERATH Piller Blowers & Compressors GmbH (Germany)
Richard KÖNIG Technische Universität Dresden (Germany)
Albert LANGKAMP Technische Universität Dresden (Germany)
Maik GUDE Technische Universität Dresden (Germany)

The performance and efficiency of future and existing blowers can be improved by using new types of high-performance impellers made of fibre composite materials. Composites have excellent density related mechanical properties allowing to significantly increasing the rotational speed. Reduced mass of the impellers, especially the lower inertia, allows faster reactions to variable speeds and uses less energy. Composites enable variable designs of the blades, which leads to flow-optimised cross-sections improving the efficiency and may reduce noise. Furthermore, the layered structure of such impellers for the integration of sensors provides a basis for component monitoring and data generation.

Due to the large number of adjustable parameters, the special knowledge required for their design as well as the high expenses and risks for the development and production of integral rotors, these are currently only niche products.

Therefore, in cooperation with the German Forschungsvereinigung für Luft- und Trocknungstechnik (FLT), a modular design of a high-performance impeller was developed at the TU Dresden, whose functionality was presented in a first FAN paper [1]. Compared to the integral solutions, the manufacturing effort of this modular design is significantly reduced, allowing higher availability and variability in production. In addition to the developed design, a numerically based methodology for the mechanical construction was elaborated and presented and allows to consider the material specific damage and failure behaviour. Using this virtual development, speeds of approx. 7,500 rpm were achieved in the first test, corresponding to a circumferential speed of approx. 400 m/s and increasing the speed by approx. 30% compared to the metallic model.

Using the developed and manufactured fans, this paper shows the potential of such modular fibre composite-metal hybrid structures for vibration-damped fans. In a first step, numerical and experimental modal analyses for the determination of natural frequencies, vibration modes and modal loss factors on disc rotors made of steel and fibre composite material were performed and compared. Subsequently, a fibre composite impeller was manufactured and also subjected to an experimental modal analysis. The determined loss factors impressively confirm the high potential of fibre composite materials for vibration damping of impellers. The fibre-composite metal hybrid construction method mentioned above also enables a further improvement in vibration damping through additional joining zones.

[1] Spitzer S., Pohl M., Grothe R., Langkamp A., Hermerath P., Gude M.: Endless fibre-reinforced composite-metal-impeller: material related design and dimensioning process for hybrid radial-fans. FAN2018, Darmstadt, 18-23. April 2018

Keywords: Composite Material, Radial Fan, Hybrid Design, Steel, Damping

martin.pohl@tu-dresden.de

Evaluation of the Wear-Resistant Plate Performance on Different Locations Over the Flow Path of a Large-Sized Heavy-Duty Centrifugal Fan

Nicola ALDI	Università degli Studi di Ferrara (Italy)
Nicola CASARI	Università degli Studi di Ferrara (Italy)
Michele PINELLI	Università degli Studi di Ferrara (Italy)
Alessio SUMAN	Università degli Studi di Ferrara (Italy)
Alessandro VULPIO	Università degli Studi di Ferrara (Italy)
Stefano OLIANI	Università degli Studi di Ferrara (Italy)
Nicola ZANINI	Università degli Studi di Ferrara (Italy)
Paolo SACCENTI	Boldrocchi srl (Italy)

In industrial applications as chemical plants, cement factories, and glassmakers, large-sized centrifugal fans are commonly used for dust-laden flow processing. In many cases, the contamination is due to solid particles which are responsible for fouling and erosion issues. Erosion induces the reduction of mechanical resistance and at the same time, the modification of the geometry and the surface characteristics of the internal flow path. The process works according to the characteristics of the erodent particles, such as dimension and hardness which have to be coupled with the mechanical characteristics of the substrate, like hardness and roughness level. In addition to this, the intensity of the erosion depends on the dynamic characteristics of particles, especially velocity and impact angle. For these reasons, erosion-related issues are difficult to be predicted and reduced. In the attempt of preserving the structural integrity of the internal walls, wear-resistant plates are positioned where the impacting contaminants are supposed to be more detrimental. In the present work, a combined experimental and numerical approach is proposed to evaluate the proper set up of wear-resistance plates over the flow path of a large-sized centrifugal fan. The results show how different regions (rotating and stationary walls) are subjected to different impact behavior and for this reason, the design of the position of the wear-resistant plate is not straightforward. Suggestions related to the reduction of the erosion intensity are reported, highlighting the possibility to design the best compromise between erosion, performance, and costs.

nicola.aldi@unife.it

INFLUENCE OF WINGLETS ON THE TIP VORTEX OF LOW PRESSURE AXIAL FANS

Daniel KESSLER Technische Universität Braunschweig (Germany)
Jens FRIEDRICHS Technische Universität Braunschweig (Germany)
Frieder LÖRCHER ZIEHL-ABEGG SE (Germany)

Low-pressure axial fans serve a wide range of applications and combine the requirements for high flow rates and low-pressure rises. Due to the low Reynolds numbers and low workload of the fluid, the design of low-pressure axial fans differs considerably from compressors in gas turbines. Yet, regulations and customer demand for high efficiency and acoustic standards. A primary goal of the ventilation and air conditioning industry and the automotive industry is to improve the fans' acoustic. Therefore, a lower overall sound power level and tonal components are necessary. One of the sources for a high sound power level is the interaction of the tip vortex with the main flow through the fan or other fan blades. The operating point essentially defines the tip vortex trajectory. Yet, the sweep and shape of the blade tip geometry can influence the bursting of the vortex and its size.

The aim of this paper is the investigation of three different blade tip geometries or winglets. By using basic designs, the fundamental influence on the behavior of the tip vortex and the acoustics can be shown. Aerodynamic (including characteristic and wall pressure measurements) and acoustic studies were carried out. The results of the different blade tip geometries are compared to a well-research baseline fan. A qualitative correlation between the local design of the blade tip and the effects on the tip vortex on the one hand and the acoustics, on the other hand, could be shown. A three-dimensional design using winglets leads to a more stable and shifted vortex. A reduction of the overall sound power level in overload and at the design point is shown. Yet, in partial load, the vortex is much more diffuse, which leads to increased broadband noise.

d.kessler@ifas.tu-braunschweig.de

EFFECT OF INLET GAP CONFIGURATION ON TONAL NOISE FROM CENTRIFUGAL FAN

Martin OTTERSTEN Swegon operations / Chalmers University of Technology (Sweden)
Hua-Dong YAO Chalmers University of Technology (Sweden)
Lars DAVIDSON Chalmers University of Technology (Sweden)

Heating, ventilating, and air conditioning systems (HVAC) are today an important part of many people's life. They provide a sufficient amount of airflow with the correct temperature, quality, and humidity. The negative side is the noise it produces. Modern HVAC systems are driven by low-speed isolated centrifugal fans which produce the dominant tonal noises. There are no obstacles that the blades pass in this type of fan. However, the dominant tonal noise produced by the fan is at the blade passing frequency (BPF). This type of fan has a gap between the rotating shroud and the stationary inlet duct. Previous studies have shown that the flow through the gap causes turbulence that develops along the shroud wall and interacts with the blades at their leading edge. The interaction renders uneven surface pressure distributions among the blades as well as significant peak differences. The location of the tonal noise sources for BPF agrees with the locations of these interactions and surface pressure peaks. This study aims to investigate how the tonal frequencies are affected for different gap configurations. Two new designs are compared with a reference fan that was used in previous studies. The approach is to use the hybrid computational aeroacoustics (CAA) method, that couples the improved delayed detached eddy simulation (IDDES) method with the Ffowcs Williams and Hawkings (FW-H) equations. Our simulations show that the gap design effects the magnitude of the tonal frequency, BPF. By increasing the size of the gap, the amplitude can be decreased. The acoustic sources for the BPF agree with the location of the uneven surface pressure distribution on the blade leading edge. The time history of the surface pressure is therefore compared and it is shown that the gap configuration effects the amplitude of the pressure fluctuations. Also, the fan performance is affected by the gap configuration. The static pressure rise made by the fan and the pressure upstream of the fan in the inlet duct is influenced by the gap design.

martin.ottersten@swegon.se

EFFECT OF TIP CLEARANCE ON ACOUSTIC AND AERODYNAMIC PERFORMANCES

Patrice PÉPIN École de Technologie Supérieure (Canada)
Marlène SANJOSÉ École de Technologie Supérieure (Canada)

New fan concepts must achieve high efficiency and low noise levels in order to offer relevant alternatives in an increasingly competitive market. Analytical tools based on simplified theoretical approaches allow to quickly propose interesting concepts. Numerical simulation in fluid mechanics is used to complete these approaches because it offers a three-dimensional understanding of the flow by considering, to some extent, the effects of secondary flows.

In this study, various numerical simulations are performed to analyze the tip clearance impact on a low-speed axial fan performance and noise emission. The studied configuration is the USI7 installed in ducts and experimentally tested at the University of Siegen. Initially, the work focuses on calibrating the mesh (on the casing, the blades, the clearance) and the calculation parameters in order to reproduce the aerodynamic performance of the fan at its nominal speed (3000 rpm) with a clearance corresponding to 1 % of the nominal impeller diameter. This configuration is then used to perform simulations on the same fan with a thinner clearance of 0.1 % of diameter. Afterwards, systematic analyses of the flow are carried out on the two configurations in order to quantify the performance losses associated to the increased clearance and to evaluate if acoustic models based on steady-state RANS calculations can accurately evaluate the noise emission. The results show that the accuracy of RANS simulations is limited for large tip clearance due to unsteady flow mechanisms. The noise predictions suffer from missing modeling for these complex flow features. Still the aeroacoustic analytical noise models allow to predict the noise increase associated with tip clearance increase.

pepin.patrice22@gmail.com

Aerodynamic Optimization of a Low-Pressure Axial Fan using Adjoint Computational Fluid Dynamics

Konrad BAMBERGER University of Siegen (Germany)
Thomas CAROLUS Steinbeis-Transferzentrum Strömungstechnik und Strömungsmaschinen (Germany)

The presentation discusses the aerodynamic optimization of an axial fan using adjoint Computational Fluid Dynamics (CFD). As an example it is tried to improve the aerodynamic performance of the University-of-Siegen reference low pressure, rotor-only axial fan USI7. The existing OpenFOAM solver "adjointShapeOptimizationFoam" is customized since the original code does not account for rotating frames of reference. Moreover, a new boundary condition is implemented since in adjoint simulations, the case-specific objective function has an impact on the boundary condition. Together, the solutions of the primal and adjoint simulations yield a sensitivity map which indicates weather material should be added or removed in each cell of the numerical grid. This information is interpreted to alter the fan geometry. The new geometry is then again simulated by the adjoint code and further improved. At the fourth iteration, no further improvement can be observed. The overall CFD-predicted improvements are an increase of the total-to-static efficiency at the fan design point of + 0.5 percentage points and a delay of the stall point towards smaller volume flow rates.

The obtained new fan rotor (USI8) is manufactured via CNC-milling with high accuracy. Its performance characteristics are determined experimentally on a chamber test rig. The experiments confirm the CFD-predicted improvement at the design point, the delay of the stall point, however, cannot be validated.

The improvement of efficiency as such proves that the adjoint method is working. The fact that only four relatively inexpensive RANS simulations were necessary to obtain an improvement confirms the extremely high performance of the adjoint method as compared to other optimization methods. However, it must also be acknowledged that an improvement of 0.5 percentage points of total-to-static efficiency is very small. Reasons such as a non-optimal interpretation of the adjoint results, uncertainties of the CFD model (especially in the near stall region) or simply the too good reference fan USI7 without significant potential for improvement are addressed.

konrad.bamberger@web.de

Modern Design of Automotive Axial Cooling Fans Using Inverse Methods

Mihai BLEIZIFFER Hanon Systems EFP Deutschland GmbH (Germany)
Anand SIVARAMAKRISHNAN Hanon Systems EFP Deutschland GmbH (Germany)
Rainer SCHOELE Hanon Systems EFP Deutschland GmbH (Germany)
Mehrdad ZANGENEH University College London (United Kingdom)
Jiangnan ZHANG Advanced Design Technology, Ltd. (United Kingdom)

Many companies developing automotive axial fans have database of fan geometries that have been developed over a number of years as a result of long period of iterative development based on CFD and experimental tests. The know-how embedded in these optimum geometries is not very transferrable to other applications where different flow rates or pressure rise are required. Also the demand for more efficient and silent devices, especially for future BEVs and FCEVs push the aerodynamicist towards the limits of the classic methodologies.

The aim of this paper is to present a methodology in which the important know-how of an optimum existing design is captured by using an inverse design method to reverse engineer the existing blade and hence recover the optimum loading of the fan blade. Then use this optimum loading as a basis for the design optimization of the automotive cooling fan that meets the new challenging requirements. In this paper, we want to show how this approach was implemented as well as the benefits gained on performance, acoustics and time-to-delivery.

In order to combine both existing methods a complex aerodynamic design process involving reverse engineering, CFD simulations and experimental validation has been set. Starting from the loading obtained from reverse engineering of an existing design a process was followed in which the spanwise work and streamwise loading were varied systematically by using the inverse design method and the resulting performance evaluated by using a previously validated CFD simulation set up. Parallel, structural and modal analysis simulations were performed to ensure the integrity during operation. Performance measurements have confirmed the CFD simulations results and acoustic measurements showed an excellent behavior.

mbleizif@hanonsystems.com

MULTI-POINT, MULTI-OBJECTIVE OPTIMIZATION OF CENTRIFUGAL FANS BY 3D INVERSE DESIGN METHOD

Mehrdad ZANGENEH University College London (United Kingdom)
Jiangnan ZHANG Advanced design Technology Ltd (United Kingdom)

Centrifugal fan stages are used in many applications where relatively high pressure rise is required in compact size. The application can vary from household appliances, industrial to airconditioning and data centre cooling applications. In many of these applications the fan stages are required to meet multi-point requirements in terms of both pressure rise and efficiency.

In this paper we present the design of a centrifugal fan stage for multi-point and multi-objective requirements. The paper starts from basic requirements for the design such as pressure rise, flow rate and rpm at 2 operating points. The fan stage needs to meet a maximum torque requirement set by the motor power. An initial flow path for the stage is generated by using a meanline code by using one of these operating points. This meridional geometry is then used in a 3D inverse design method in which the 3D blade geometry of the impeller is generated for a given distribution of loading (or pressure jump) on the blade. The aim of this initial design is to create a baseline for multi-point optimization. For optimization the meridional geometry of the impeller and inlet nozzle are parametrized. The blade loading is also parametrized on one streamwise location in order to meet the requirements for the blade to be manufactured by a metal pressing process. In total 14 parameters are used for meridional shape and blade shape.

By using a wide range of variation of these 14 design parameters and Design of Experiments method a design matrix is generated for about 120 fan geometries. These geometries are then run in steady 3D RANS code for the two operating points. The CFD set up used tries to represent the measurement set up and hence covers a suitably large inblock and outblock boundary. Important objectives such as efficiency, pressure rise and torque are then extracted from the CFD solution at the two operating points. These results are then used together with the design matrix in a surrogate model based Kriging method. A multi-objective Genetic Algorithm (MOGA) is then run on the surrogate model to find trade offs between the efficiency and pressure rise at the two operating points subject to constraints on maximum torque and also slope of the pressure rise change between the two operating points. The design with the best efficiency at the two operating points is then exported from the surrogate model and run in CFD. The CFD results confirm significant improvement in efficiency over the baseline design and also show that the actual CFD values for efficiency, torque and pressure rise at the two operating points are very close that predicted by surrogate model. Hence confirming the accuracy of the surrogate model used for design optimization. This process can be used to speed up the design optimization of centrifugal fan for multiple operating points under industrial time scales.

m.zangeneh@ucl.ac.uk

OPTIMIZATION, CONTROL AND DESIGN OF ARBITRARILY SHAPED FAN ARRAYS

Daniel CONRAD ebm-papst Mulfingen GmbH & Co. KG (Germany)
Jonathan MAYER ebm-papst Mulfingen GmbH & Co. KG (Germany)
Erik REICHERT ebm-papst Mulfingen GmbH & Co. KG (Germany)

Nowadays, medium to large sized fans in traditional applications are increasingly replaced by fanarrays, that is, a number of usually smaller fans operating in parallel. Although the aerodynamic efficiency of single fans still rises through further development, the research and development the field of fan systems will get even more important in the future. Besides redundancy, these fan arrays offer additional degrees of freedom. In applications with varying operating points, high improvement in system efficiency and high energy savings, respectively, can be achieved by selectively turning off some of the fans in an array and controlling the operation of the remaining active fans accordingly. We propose a method for the optimal control of such fan arrays.

In our approach, the arrays can be composed of an arbitrarily number of differed sized fans and/or fans with different fan curve characteristics. There are only two prerequisites present in the theoretical derivation of the method. It is assumed that the speed of each fan in the array of fans can be variably controlled up to its maximum rotational speed. Moreover, it is assumed that no backflow occurs in situations where fans are turned off. In reality, this can be achieved using, i.e., some kind of shutters or flaps. For a given fan array and operating point, the method yields the optimal number and combination of active fans and their optimal rotational speed, which maximizes the system efficiency. Lastly, applying this method to the inverse problem, where the shape of the actual array is yet unknown yields an optimal design of fan arrays for given device and fan size constraints.

daniel.conrad@de.ebmpapst.com

OPPORTUNITIES TO UPGRADE: THE EFFECT OF A FAN FAULT IN AN AIR HANDLING UNIT FAN GRID

Daniel HOPKINS ebm-papst UK Ltd (United Kingdom)
Andrew BURGOYNE ebm-papst UK Ltd (United Kingdom)

With climate change upon us, reducing our carbon footprint is more important than ever. Across the non-residential building sector, owners and occupiers are looking at building services equipment within their estates, to exploit running cost savings, reduce carbon footprint and smarten up using Internet of Things (IOT) technology. One area being refreshed is Air Handling Units (AHU's) that provide fresh air, remove stale air, heat, cool and control humidity.

Many buildings operational today will still be operational for years to come and are installed with legacy air handling units using AC induction motor fans, commissioned to deliver the same performance every day. Recent efforts to deliver energy savings via demand control has seen installation of inverter drives to vary the performance of the fans via speed control. The cumulation of impeller aerodynamic efficiency, the efficiency of the belt and pulley transmission, motor efficiency and inverter drive efficiency provide an opportunity optimise air movement. In addition, a legacy AHU that employs one fan per supply or extract flow path that has a breakdown during service, has no back-up.

Existing AHU's due for refurbishment are increasingly being upgraded to use multiple Electronically Commutated plug fans arranged in a grid. Sharing the required airflow across multiple fans allows flexibility in selecting fan diameter, number of fans in the grid and provide n+1 redundancy. In addition, fitting inlet rings with a flow measurement pressure tapping can monitor, control, and adjust the fan grid behaviour to deliver the required flowrate using a constant volume control system.

But what happens if one of the fans fail?

In a single fan AHU, if there is no functioning fan there is no airflow, in a multiple fan system one failure does not stop the delivery of air.

This paper explores a number of aspects of Fan Grid installation in Air Handling Units:

1) The accuracy of the inlet ring volume flow measurement system in comparison to an airflow test rig designed to ISO5801 in normal operation;

2) The effect on the accuracy of the flow measurement system in the event of fan failure when air can recirculate through the faulty fan;

3) The effect on the accuracy of the flow measurement system in the event of fan failure when a back-draught damper (Gravity Shutter) is fitted to prevent air recirculation.

dan.hopkins@uk.ebmpapst.com

SUSTAINABLE BUILDING VENTILATION BY ALGORITHMICALLY SUPPORTED DESIGN OF SEMI-CENTRAL FAN SYSTEMS

Julius BREUER Technische Universität Darmstadt (Germany)
Tim Moritz MÜLLER Technische Universität Darmstadt (Germany)
Peter Franz PELZ Technische Universität Darmstadt (Germany)

Mechanical ventilation systems are becoming more and more necessary in increasingly airtight buildings and they can contribute to high energy efficiency when used in combination with heat recovery systems. The prerequisite for this is that the energy consumption of the fans is low. This can be reached by (i) reducing the required hydraulic power and (ii) making the conversion as efficient as possible. One approach to decrease the hydraulic power (i) is the decentralized placement of fans in a central duct network (so called semi-central systems) to avoid throttling losses. To increase the efficiency (ii), the interaction of components must be balanced. Especially with fluctuating loads and complex topologies, countless variants have to be compared and evaluated for this purpose. In order to still master the combinatorial explosion, algorithm-based design methods from the field of discrete mathematics are a promising approach.

Therefore, in this contribution we present an approach to design semi-central ventilation systems for non-residential buildings using discrete optimization methods. We first present the methodology to select the placement of the decentralized fans based on an assessment of the hydraulic power and throttling losses. Then we present the techno-economic model of the components and the system. Based on this, the design task – the selection, interconnection and operation of the fans – is formulated as a mixed-integer nonlinear optimization problem. Here, the life-cycle costs are minimized considering different volume flow demands due to the occupancy in the rooms. For the solution, exact optimization methods from the field of discrete optimization are used, which have the capability to ensure the global optimality of the solution. This is applied in a case study, in which we optimize the ventilation system of a building of the Technical University of Darmstadt. Finally, the results are discussed and the potential of our approach for cost and energy savings is demonstrated.

julius.breuer@fst.tu-darmstadt.de

SOURCE LOCALISATION ON LOW SPEED AUTOMOTIVE FAN

Yann PASCO Université de Sherbrooke (Canada)
Gyuzel YAKHINA Université de Sherbrooke (Canada)
Stéphane MOREAU Université de Sherbrooke (Canada)

The sound generated by rotating machines is a topic that is more and more considered. In fact, this noise is an environmental concern in many industrial fields. The rotating machine studied here is an automotive engine cooling fan system. The noise produced by these fans is a major contributor to the total radiated noise and in certain cases it dominates other sources like the engine or the tires. Usually, the fan is extracting the air through a heat exchanger where a coolant fluid is circulating through a network of tubing. The heat exchanger is usually located close to the fan on its suction side. This creates a turbulent flow and participates to the generation of broadband noise. As the module is located in the engine bay, the upstream flow is distorted which leads to a strong tonal noise contribution. The impact of those distortion on the blades or struts generates tonal noise.

The article aims to evaluate the contribution of acoustic sources located on the rotor blades. The source localization for rotating machine should take into account the Doppler effect as the sources are rotating with the fan. Classical beamforming works but it leads to continuous annulus sources. Sitjsma et al. [1] propose to solve this problem by using a transfer function for a moving monopole source in uniform flow to reconstruct the signal in the time domain and develop an algorithm able to deal with rotating sources, named ROtating Source Identifier (ROSI). Using this method, the measurement of the sound pressure of rotating sources can be done using an array of microphones. The proper implementation of the method was verified using the benchmark from [2]. In the present study, a 60-microphones logarithmic array was used and the reconstruction of sound pressure level was done in third octave bands. Measurements were done in different rotational velocity of the fan in the anechoic chamber of Sherbrooke's university (UdeS) without any other flow restriction than the heat exchanger itself. A measurement was done at full RPM with and without heat exchanger to compare if it is transparent acoustically. The Directivity measurements were conducted as well. Both method has been compared to the central microphone of the microphone array.

The present paper is a part of a collaborative project between UdeS and Von Karman Institute (VKI). The major aim is to conduct an experimental investigation of the fan noise in two different facilities and to compare the results.

[1] P, Sijtsma and S, Oerlemans and H, Holthusen. "Location of rotating sources by phased array measurements", AIAA 2001-2167, 2001

[2] Herold, G. "Microphone Array Benchmark b11: rotating points sources", 2017

yann.pasco@usherbrooke.ca

MULTI-MODAL ACOUSTIC POWER CHARACTERISATION OF DUCTED ELEMENTS

Yackine TADJOU Centre de Recherche et d'Innovation Technique et Technologique en Moteur et Acoustique Automobile (CRITT M2A) / Université de Technologie de Compiègne (France)
Jean-Daniel CHAZOT Université de Technologie de Compiègne (France)
Emmanuel PERREY-DEBAIN Université de Technologie de Compiègne (France)
Arnaud PLESSY Centre de Recherche et d'Innovation Technique et Technologique en Moteur et Acoustique Automobile (CRITT M2A) (France)

Methods and guidelines for acoustic source power characterization of elements in circular ducts with flow are presented. Acoustic pressure measured by microphones is used as source data for this study conducted above plane wave propagation. Thus, a multi modal approach is considered including azimuthal and radial modes for forward and backward directions. These in-duct propagating modes are decomposed from the acoustic pressure field with the matrix of modes evaluated at the positions of the microphones for the entire frequency range considered. Mode amplitudes are then obtained by inversing this matrix and used to obtain acoustic power of propagating modes. Because of the non-intrusiveness constraint, the conditioning of the microphone-mode matrix can be bad. It is minimized by optimizing microphone positions with a Genetic Algorithm (GA). Satisfactory results are obtained for the entire frequency range considered.

To validate the modal decomposition, an analytic model is developed. Straight cylindrical ducts closed with a rigid wall on an end and porous materials on the other modelized with a Johnson-Champoux-Allard (JCA) model are considered. Between both extremities, a plane acoustic source is placed on the duct and further a measurement section with the optimised positions of microphones. Amplitudes of propagating modes are calculated in the air domain and compared to the ones calculated from acoustic pressure measurement on a test-bench.

A parametric study is conducted to evaluate the impact of the uncertainty induced by the measurement chain but also from the turbulent boundary layer noise with a Corcos model on the modal acoustic power. The acoustic power for each propagating mode in an infinite and rigid cylindrical duct is calculated from synthesized data with and without perturbation. The study shows that the measurement chain uncertainty has small impact on the calculated acoustic power. A denoising method based on pressure correlation is also included in the parametric study.

ytadjou@crittm2a.com

TOWARDS A CENTRIFUGAL FAN DATABASE FOR OFF-DESIGN VALIDATION OF CFD SIMULATIONS

Matthias PROBST Karlsruhe Institute of Technology (Germany)
Balazs PRITZ Karlsruhe Institute of Technology (Germany)
Hans-Jörg BAUER Karlsruhe Institute of Technology (Germany)

Turbomachineries such as fans are energy conversion machines which can be found in many daily technical devices. The bandwidth of power consumption ranges from a few watts to megawatts and energy-saving requirements are getting increasingly strict with every year. Therefore, a continuous study of the flow phenomena inside such machines is necessary to identify and to understand the sources of energy losses. To analyze the flow phenomena beyond empirical findings, Computational Fluid Dynamic (CFD) simulations are nowadays being used to unlock further potentials for design optimizations and energy savings without the need for expensive experiments. While the optimal operation point typically can be described well by such simulations, the off-design conditions of turbomachineries are very challenging to predict due to unsteady effects for instance.

A validation test bench for a low-pressure centrifugal fan with a spiral casing has been built and Particle Image Velocimetry (PIV) is used to measure the flow field in several different positions and various operation points. The acquired data together with the geometry and a structured CFD mesh is provided through an open-source database and is intended to be used by researchers to validate their CFD codes and methods.

On the pathway towards the centrifugal fan database, research and meta data management along with an uncomplicated data access organization are two key building blocks. This paper highlights the concept and implementation of the data management using HDF5 as the underlying data file format. Furthermore, the workflow and tools provided for other scientists, that aim to validate their fan CFD simulations, is presented.

matthias.probst@kit.edu

REYNOLDS NUMBER SCALING OF AXIAL FLOW FANS

Adam VENTER Stellenbosch University (South Africa)
Gerhardus Marthinus STEENKAMP Stellenbosch University (South Africa)
Chris MEYER Stellenbosch University (South Africa)
Johan VAN DER SPUY Stellenbosch University (South Africa)

There exists uncertainty about the accuracy with which the fan scaling laws can predict the performance characteristics of a full scale prototype fan from model testing. This uncertainty stems from the large difference in the value for Reynolds number that typically exists between the prototype and model fan. Increasing the turbulence intensity of the inlet flow during testing of the model fan is a suggested method for improving Reynolds number similarity between the two scenarios.

The M-fan was designed as a low pressure rise, high volume flow rate fan, for application in a specific Concentrated Solar Power (CSP) plant. It has been shown to exhibit two dimensional flow across its blades far from the end walls. Assuming that the M-fan's solidity is low enough so that the blades can act as isolated aerofoils, allows for a comparison between the effect of increased free stream turbulence intensity on the performance of the M-fan and the lift and drag characteristics of a two dimensional aerofoil. The similarity between increasing the Reynolds number of the flow and increasing the turbulence intensity of the flow is demonstrated on a NASA LS 0413 aerofoil, used in the blading of the M-fan, through two dimensional aerofoil testing in a wind tunnel. Increased free stream turbulence intensity is shown to result in a clear delay in stall whilst increasing the maximum coefficient of lift, with the effect becoming more prominent as the turbulence intensity increases.

Results from model testing of the M-fan at its design, and moderately increased, blade setting angles is shown to be independent from any increase in inlet flow turbulence intensity. Thus, the fan test results are shown to be Reynolds number independent. Model testing at a very high blade setting angle showed a steady increase in fan static efficiency with an increase in turbulence intensity. This is similar to increasing the Reynolds number at which the fan operates, thus validating the experiment. These results indicate that, at the scale test of 1.542 m diameter and 720 rpm, the M-fan operates independent of the Reynolds number. The use of the fan scaling laws on results obtained from model testing of the M-fan under these conditions will yield accurate results when scaled to represent the fan characteristics of the full scale prototype fan.

wiehansteenkamp@gmail.com

Development of Computational Methods for Inlet Optimisation in Centrifugal Ventilation Units

Daniel OWEN Elta Fans Ltd (United Kingdom)
Luke TEMPLE Elta Fans Ltd (United Kingdom)

Legislation introduced to increase the efficiency of energy-using products, such as fans and ventilation units offered to market presents ever greater challenges in the development of products that meet both minimum efficiency requirements and the needs and expectations of customers. Airflow performance and efficiency of a ventilation unit can be improved though careful consideration of the path the conveyed air takes through a housing or enclosure, and suitably guiding the air to reduce internal pressure losses.

Physical performance test iterations to optimise air path geometry are time consuming and require significant materials. Optimisation of air path geometry through computational fluid dynamics (CFD) reduces the number of physical test iterations and introduces a greater control over the test environment; however, this requires a certain level of machine capability, operator skill and understanding to sufficiently set up and review results.

This project consider factors affecting performance and efficiency in HVAC units and embarks on the development of a simple spreadsheet-based tool to predict performance, facilitating optimisation of inlet conditions. This tool is developed through benchmarking a ventilation unit with 315mm inlet spigot, comprising an enclosure or rectangular cross section and a single backward curved centrifugal EC motorised impeller mounted on an inclined bulkhead (of known performance characteristics). A series of CFD studies are run to determine which of the key variables (such as height-width of cross section & degree of bulkhead incline) affect the airflow performance and efficiency of the ventilation unit and from this a parametric optimisation of upper and lower variable limits defined.

The result of these studies are used to generate a series of enclosure loss polynomial curves which are then developed into a spreadsheet-based tool for future performance estimation against various scenarios.

A potential benefit of this tool is the reduction in physical prototype and computational fluid dynamics (CFD) iterations required in the design of new product ranges, allowing for quicker development cycles.

dowen@eltafans.co.uk

ENERGY AND CO2-IMPACT OF ECODESIGN REGULATION FOR FANS ON THE EUROPEAN MARKET

Claus HAENDEL European Ventilation Industry Association EVIA (Belgium)

Fans are omnipresent in different products. Since 2011 Ecodesign regulates minimum energy performance of fans above 125W. A fan in that meaning typically is not a final product, but used as a component in different application. From the very beginning, questions have been raised, weather it is helpful to regulate a component and not a final product.

The presentation will analyse the European market transformation from the beginning of the regulation and will estimate the energy savings achieved. Additionally it will estimate the share of fans placed on the European Market in other Ecodesign regulated products (ventilation units, air heating/cooling devices, chillers, RAC, refrigeration) and their relative impact in the energy reduction and will give a short outlook for the next steps. Furthermore a cost-benefit analysis for the estimated market impact will calculate average CO2-Reduction cost for fans.

claus.haendel@evia.eu

EXERGY BASED EFFICIENCY ASSESSMENT FOR FANS IN COMPARISON TO COMMON USED EFFICIENCY DEFINITIONS

Johannes BRÖTZ TU Darmstadt FST (Germany)
Christian SCHÄNZLE TU Darmstadt FST (Germany)
Peter F. PELZ TU Darmstadt FST (Germany)

The efficiency definition allows the comparison of two machines with each other. In general, the efficiency is defined as the ratio of usable power to the required power. This raises the question: what is the usable power? Most engineers discuss efficiency on grounds of the energy balance, i.e. the first law of thermodynamics. In this paper, we discuss the benefit of the machine and the effort to reach this benefit. We derive the exegetic efficiency taking the second law of thermodynamics into account, too. For this most general definition of the efficiency, we will see, that both axioms are indeed needed. As a result, its biggest benefit is thus its universal applicability. The exergetic efficiency does not only consider the fluid power from pressure build-up and delivery of a volume flow, but also the contained usable thermal energy to environment. The function of a fan or compressor is to increase the exergy of the delivered gas. The exergy measures the working capacity of the gas relative to its environment. The derivation shows two ways to calculate the exergetic efficiency, but only one is suitable for the application. For a caloric and thermal ideal gas the exergetic efficiency can be calculated without further effort in comparison to standardized performance testing.

On this basis, a comparison between isentropic and exergetic efficiency is given. A high-pressure radial fan is used as an example and the differences between both efficiencies are discussed. Therefore, measurements at a non-adiabatic fan is evaluated and the role of the heat flux to the environment is discussed. For a standard such as ISO 5801 "*Fans - Performance testing using standardized airways*", efficiency must not only be physically correct. It must also be simple and practical. Against this background, the outlook of this paper discusses when and which efficiency definition is appropriate and best suited for a standard.

Johannes.Broetz@fst.tu-darmstadt.de

European Fan Efficiency Standard - Plans for Revision Including Standardised Product Information at Partial Load to Improve Fan Application

Geoff LOCKWOOD European Ventilation Industry Association (Belgium)

The efficiency test standard FprEN 17166 "*Fans - Procedures and methods to determine the energy efficiency for the electric input power range of 125 W to 500 kW*" was developed at the request of the European Commission. It was requested to support regulation 327/2011 ecodesign requirements for fans driven by motors with electric input power between 125 W and 500 kW. In drafting the standard, technical experts of the technical committee CEN TC156 WG17 took the opportunity to create new terms and definitions to clarify issues that occurred with the publication of the regulation. But some aspects had to follow the legally binding requirements of the regulation leaving unresolved matters. The request in the European Commission annual Union work programme for European standardisation for 2021 gives an opportunity to put right those problems and provides an opportunity to further improve the system approach. The industry and other stakeholders should carefully consider what needs to be in the Standardisation Request (sReq) that will be the basis of the Commission's instruction to CEN TC156 WG17. The European Ventilation Industry Association (EVIA) is being proactive with a proposal.

Prior to FprEN 17166, significant elements of the fan that are necessary to determine the performance were taken for granted. Fan engineers knew what was required, but not every element was defined. Some terms and definitions were missing from standards, and others unexpectedly interpretation. This resulted in misunderstanding and products being within the scope of the regulation, but without methods to determine the energy performance.

The draft standard is close to being a desired system approach. A revision can put in place the requirements to provide integrators and users with information to make better informed decisions. Those requirements can include a common method to determine, publish and interpolate data across the complete operating range including partial load.

Standards for motors and VSD have, and are in the process of, being revised to provide methodologies to determine the performance and losses at any operating point. A revision of the fan standard will provide an opportunity to incorporate those methods and to define when calculation methods are appropriate.

The Fan working group of EVIA has defined the problems of aspects of the ecodesign fan regulation, listened to the requests of other stakeholders for revised standardisation and has made a proposal for change to the Commission. This paper describes those proposals.

geoff.lockwood@uk.ebmpapst.com

Effect of the Fan-Heat Exchanger Interaction on the Acoustic Emissions of Heat Pumps: Experimental and Numerical Studies

Sebastian WAGNER Fraunhofer Institute for Building Physics IBP (Germany)
Jens ROHLFING Fraunhofer Institute for Building Physics IBP (Germany)

In recent years research on heat pump technology has resulted in systems with high energetic efficiency. Therefore modern, electrically driven heat pumps have become an attractive alternative to conventional heating systems. Today an increasing number of heat pumps are installed in dense urban areas. Hence, the noise emissions from heat pumps are becoming increasingly relevant. Acoustic research on heat pumps with respect to noise control and sound perception has become a major topic in the research and development efforts of heat pump manufacturers.

Aeroacoustic noise from fans can contribute significantly to the overall noise emissions from outdoor units of heat pumps. The fan is used to move ambient air across a heat exchanger, which transfers the thermal energy from the air to the refrigerant. There are various heat exchanger concepts and possible fan - heat exchanger arrangements, which can be used when designing such an outdoor unit.

To date there has only been little research on how different types of heat exchangers and their relative arrangement with the fan effects the acoustic emissions of heat pumps. In order to investigate the effect of the fan – heat exchanger interaction on the acoustic emissions, experimental studies have been conducted on a simplified test setup. The set-up comprises a fan and heat exchanger in an adjustable casing, closely resembling the outdoor unit of a heat pump. This experimental setup allows the systematic investigation of various fan – heat exchanger configurations, e.g. variations of distances and angles, as well as different types of heat exchangers, such as classical round tube and microchannel heat exchangers. In parallel to the experimental studies, CFD and CAA simulations have been conducted in OpenFoam®. In the simulations a simplified geometrical model of the experimental set-up is used to analyse the air flow through the heat exchanger – fan assembly. The numerical model is based on an incompressible CFD simulation, from which the sound pressure in the far field is calculated using the Ffowcs Williams and Hawkings approach.

This paper presents experimental and numerical results for various fan - heat exchanger configurations for a classical round tube heat exchanger and a microchannel heat exchanger. The results show that the characteristics of the noise emissions vary for different types of heat exchangers and also depend on the distance between the heat exchanger and the fan.

sebastian.wagner@ibp.fraunhofer.de

DESIGN OF A LOW-NOISE AXIAL FAN FOR HEAT PUMP APPLICATIONS

Martina BOLOGNINI Elica spa (Italy)
Stefano TORREGIANI Elica spa (Italy)
Giuseppe BRUNI Fime (Italy)
Savino PONTINO EMC FIME (Italy)

In this paper the design of an axial fan for a heat pump is presented. This technology is experiencing a big growth in the number of applications in the residential heating sector and investments are expected to keep increasing in the near future. Heat pumps are characterised by the exploitation of thermal energy from the external environment and represent a green alternative to traditional fossil fuel systems capable of reducing CO2 emissions. In recent years research has mainly focused on maximising the efficiency of these products. However, nowadays the effort of manufacturers must shift to improving acoustic performance, a feature that is increasingly demanded by the market.

Therefore, the designed axial fan must not only have a high efficiency in the desired working points but also low noise levels. These are the targets of the project, while the constraints are imposed by the costumers in terms of case diameter and rotation speed. An approach based on vibro-acoustic simulations has been adopted to obtain an already efficient and functional prototype. Firstly, a 2D design of the airfoil and blade has been performed in order to get to an initial fan geometry. This has then been optimized with subsequent and more accurate CFD simulations. Structural resistance has been ensured through FEA simulations and successive reinforcements.

Great attention has been paid to the study of noise which, given the particular configuration of axial heat pumps, represents a critical issue for this kind of products. Specific regulations that set limits in emissions, market analyses and an experimental benchmark on competitors' products have been considered to define the design requirements. Starting from these data and the obtained geometries a study of structure and air borne noise has been carried out. Acoustic analyses have been performed in order to investigate both tonal and broad-band noise due to vortex shedding which required transient CFD and hybrid aero-acoustic simulations to identify the main noise sources. Suitable turbulence models and solver parameters have been chosen in order to ensure high quality results and acceptable computation times. Fan resonance issues have been avoided through vibration and modal simulations for the structure of the impeller. The goodness of the results has been verified through literature study on aero-acoustic simulations, still few in number, and an experimental benchmark.

m.bolognini@elica.com

SOUND LEVEL CONTROL FOR AIR HANDLING UNITS

Ralph KRAUSE Institut für Luft- und Kältetechnik gGmbH (Germany)
Thomas OPPELT Institut für Luft- und Kältetechnik gGmbH (Germany)
Christian FRIEBE Institut für Luft- und Kältetechnik gGmbH (Germany)
Leopold MODRAKOWSKI Institut für Luft- und Kältetechnik gGmbH (Germany)
Karsten HACKESCHMIDT Institut für Luft- und Kältetechnik gGmbH (Germany)

People react very differently to the background noise of their environment depending on the activity they are performing. If the background noise is excessive, the sounds are perceived as unwanted noise. Focused intellectual work, for example, regularly requires a lower noise or background sound level than physical activity.

The background sound emission of air handling units (AHU) is also perceived as either disturbing or acceptable, depending on the situation. This is particularly problematic in rooms with highly variable human activity, i.e. rooms in which phases of focused, noise-emphasising work and physical activity alternate. An example of such spaces are school classrooms, where noise pollution - also from air handling units - has to be avoided during lessons, while background noise is likely to be submerged and negligible during break times.

The use of air handling units is often essential in order to maintain hygienic standards. In common rooms with several people in them, these devices can be perceived as disturbing during quiet periods. Therefore, there is a risk that they are switched off for the sake of convenience. However, as soon as the level of activity in the room increases, the unit is not necessarily put back into operation, which ultimately means that a pathogen-reducing air exchange then no longer exists: This situation has to be avoided.

In this presentation, a concept is introduced with the objective of suppressing or reducing the sound emission of air handling units during idle periods (without putting the air handling unit out of operation) and allowing it during active periods. The sound level emitted by the air handling units shall be adapted to a desired background noise in the respective room depending on the situation. Such sound level adjustment has long been applied to devices and systems with loudspeakers. With this technology, also known as automatic volume control, the volume of a loudspeaker is increased or decreased depending upon the volume of the ambient noise. Unlike this well-known principle, the concept presented here permits the adaptation of a noise source (e.g. AHU) to the sound level present in a room. It consists of a microphone, an analyser and a controller and can be used in all common ventilation units (mobile air purifiers, central ventilation and air conditioning systems, etc.).

The first findings will be presented at this presentation. The focus lies on the operating principle and the results of the measurements in a room with adjustable reverberation time and different positions of the noise sources and the sound sources due to the ventilation equipment.

ralph.krause@ilkdresden.de

40

INFLUENCE OF THE FAN INSTALLATION ON ITS PERFORMANCE CURVE

Alain GUEDEL CETIAT (France)
Joseph BROOKS AMCA International (United States)

Fans are tested and certified in laboratories under ideal conditions—that is, conditions intended to facilitate maximum performance. The conditions under which fans ultimately are placed into service, however, can be far from ideal. The resulting degradation in air performance can be attributed to a phenomenon known as system, or installation, effect.

At the inlet of a fan, an improperly placed fitting, such as an elbow, an inlet box, or a heat exchanger, can produce a vortex, or turbulent non-homogeneous velocity area. This less-than-optimum airflow condition leads to a reorganization of flow inside of the fan's impeller and, thus, a deterioration of the fan curve.

At the outlet of a fan, an improperly connected fitting, such as an elbow, a damper, or a duct branch, can produce non-uniform flow profiles. This degraded flow will increase pressure loss across the fitting. When an outlet obstacle is close to the fan's impeller, the flow pattern inside of the impeller may be disturbed, causing a deterioration in performance.

This presentation will show how fan performance is affected by both inlet and outlet connections. It will summarize the main points of International Organisation for Standardisation (ISO) Technical Report 16219, "Fans – System Effects and System Effect Factors," which was published in September 2020. The paper will explain the origin of system effect, present ways to quantify system effect, describe typical examples of system effect taken from a series of tests, and provide practical advice for mitigating system effect.

The paper also will discuss AMCA Publication 201, Fans and Systems, which introduced the concept of system-effect factor many years ago and is being updated with new test data.

alain.guedel@gmail.com

FE MODEL CORRELATION OF A BLISK USING A 3D SCANNING VIBROMETER

Jochen SCHELL Polytec GmbH (Germany)
Patrick BUCHWALD ITSM (Germany)
Damian VOGT ITSM (Germany)

In order to reduce weight and build more economic rotating parts, the use of bladed integral disks (Blisks) is getting more widespread. These components are milled from one single piece instead of being assembled like conventional parts.

Blisks reduce weight, but bring technical challenges. As they are made from one single piece, they show extremely low damping, leading to very sharp, pronounced vibration resonances. An ideal Blisk is perfectly symmetric. Very small imperfections in the manufacturing process lead to mistuning of the resonances. The vibration energy is then concentrated on one or a few blades, leading to higher stress in these blades and ultimately to an earlier failure of the component.

To predict the real stresses, a detailed Finite Element (FE) modelling is needed. As a first step, a FE model for a symmetric part needs to be validated by test. As a second step, it will then be adapted to account for the observed mistuning.

An experimental modal test is performed on a compressor Blisk of 240mm diameter, provided by the ITSM from the University of Stuttgart. Broadband excitation up to > 20kHz is performed by an automated modal hammer. A 3D Laser Doppler Vibrometer is used to measure the complete 3D vibration deflection shapes in a non-contact manner, leading to a data set that can easily be used for FE correlation. No mass loading or increased damping as for e.g. accelerometers appears.

Transfer functions for all scanpoints and the corresponding deflection shapes are recorded. For further analysis and comparison with FE, modal extraction is performed. These modes are then compared to the FE modes for a purely symmetrical structure. The frequencies are in excellent agreement, indicating only small mistuning. Nevertheless, the slight split in the frequencies of the symmetric mode pairs shows its presence. This is consolidated by comparing the mode shapes with a MAC analysis. While the lower order modes show excellent correlation, some of the higher order modes show lower MAC values. This indicates some localization of the vibration amplitudes to a few blades more than to others.

The used 3D Scanning Vibrometer is an excellent tool for acquiring hi-fidelity data on challenging objects like Blisks. Modal extraction of the data enables a detailed correlation with FE modes, showing slight mistuning effects in the real part. An updated FE model, which takes into account mistuning, can be set up, which is mandatory for predicting high cycle fatigue in real operation.

j.schell@polytec.de

ACTIVE NOISE REDUCTION FOR SEVERAL FANS IN TUNNEL

Sergey KARADZHI Systemair GmbH (Germany)

High efficient axial fans are widely used in tunnel ventilation and industry. Noise of these fans often becomes a crucial parameter during supplier selection because it influences people's health and safety especially in case of fire in the tunnel.

Different methods are used to reduce fans noise, but mainly with passive noise reduction – use of silencers on inlet and outlet of the fan, which in some cases can be not so efficient. In most cases, designers are using from two to four identical fans standing near each other at the same place. Apparently, this leads to an increase of sound power on 3 dB for example if two similar sources are standing near each other. However, there is an option to unphase sound signals coming from fans and reduce sound pressure levels on blade passing frequency (BPF). Sound levels on this frequency are making the main input in the total sound pressure.

For simplicity, in this work two fans were considered. To unphase signals from two sources on BPF, two options were used. First is physical change of mutual blade position for two impellers. Second is the change of mutual blade position for two impellers by changing electrical net phase. Using this method, sound levels from two fans could be reduced by up to 2 dB.

sergey.karadzhi@systemair.de

DESIGN AND DEVELOPMENT OF AN ADVANCED BESPOKE FAN SELECTION/DESIGN PROGRAM

Abd-Al Basit CHASTI WoodCock and Wilson (United Kingdom)
Ian CRUM WoodCock and Wilson (United Kingdom)
Charles HALSTEAD WoodCock and Wilson (United Kingdom)
Rakesh MISHRA WoodCock and Wilson (United Kingdom)

Due to number of challenges posed by Industry 4.0 combined with the bespoke demand of Industrial Fans owing to their wide range of applications ,there is a need for an Intelligent Fan Configurator. Currently various Industrial Fan configurators exist most of them limited to standard fan selection process with a little to no flexibility in Machine design of its components. Some of the advanced Fan selection tools such as Centrix can be used for bespoke fan selection with the capability of machine design but it lacks flexibility , cannot automate 3D Models efficiently with detail and does not have the capability for design optimisation through Inverse Design .

Fan Industries can significantly benefit from a Unified Fan configurator program that can have fan selection capabilities , flexible component design modules, detailed CAD geometry automation combined with advanced blade geometry optimization through Inverse design.

Currently such a program is being designed and developed for an Industrial Fan Manufacturing Company called Woodcock and Wilson with the help of a software platform called Drive Works. This program can be hard coded to perform various automated tasks and integrated with the widely used CAD and CAE program called Solid works.

Currently various algorithms have been devised and coded to give bespoke fan selection capabilities hand in hand with standard classical stress and noise analysis calculations. Custom Motor Selection program along with various other sales tools have also been incorporated . Base CAD models are in the process of development from where a variety of geometric needs can be automated.

Furthermore , Additional capabilities of the program will improve the performance of fan blades by giving the best base line flow path followed by the reduction in weight whilst maintaining structural integrity through inverse design.

Currently the already existing solver solving Naiver Stokes equations giving the baseline aerodynamic shape for a turbine blade will be modified to suit for fans. A new Inverse Design solver will be created to obtain best geometry for stress suitable for a given blade stress field this process will be followed by the Use Multi-Disciplinary Optimization involving machine learning and fast algorithms to come up with the final acceptable design for both Aero and Stress performance.

Combination of fast and approximate methods with slow and accurate methods hand in hand with stochastic learning from previous designs will be devised to come up with a newly optimized design selection based on only aerodynamic and Linear/Non-Linear Structural Stress Analysis.

In conclusion, this Advanced Fan configurator can open up to a variety of industrial areas for the fan companies due to its potential for executing automated CAD generation with the capability to create complex shapes that can be used hand in hand with additive manufacturing techniques to meet a specific application purpose through inverse design .

AbdAlBasit@fanmanufacturers.com

THE DESIGN AND DEVELOPMENT OF A RADON FAN FOR RESIDENTIAL BUILDINGS

Edward BENNETT Mechanical Solutions, Inc. (United States)
Artem IVASHCHENKO Mechanical Solutions, Inc. (United States)

Radon is the second leading cause of lung cancer after active smoking and the second leading cause among non-smokers. The United States Environmental Protection Agency (EPA) recommends that steps should be taken to remove Radon in homes with levels that are higher than 4 picocuries per liter. This is becoming more common as houses become more airtight to conserve energy. Existing Radon fans consist of low-efficiency vertical fans with primitive diffusers. These fans operate 24 hours per day and are not particularly efficient. The United States Department of Energy (DOE) awarded Mechanical Solutions, Inc. (MSI) a Phase 1 and 2 Small Business Innovation Research Grant to develop a new fan that would fit into the envelope of an existing Radon fan to provide a more efficient design to reduce the cost of operating the fan.

The design process started with the establishment of a manufacturing partner. Fantech and MSI entered a collaborative agreement to develop the new fan. Fantech selected a standard model to act as the source of the optimization. After that, a comprehensive CFD analysis was performed on this model to identify areas of improvement. Attention focused on the impeller and stationary diffuser. An alternate impeller geometry and two candidate diffusers were developed and optimized using a commercial turbomachinery design tool (CFturbo) and a commercial CFD code (STAR-CCM+).

MSI manufactured the fan components using additive manufacturing and constructed a test rig to test the new designs and contrast these designs with the original Fantech design. The test results conclude that both MSI prototypes outperform the original Fantech design.

MSI was required to use the original electric motor for this effort. To determine its performance, a motor test rig was built, and the motor torque and efficiency were measured. This facilitated the ability to isolate the fan aerodynamic performance. It establishes the improvement of the aerodynamics of the MSI-designed fans.

emb@mechsol.com

A Novel Axial Fan Concept Developed Using Parametric Optimization

Daniel GEBERT ebm-papst Mulfingen GmbH & Co. KG (Germany)

Axial fans are used in a wide range of applications. Therefore, when developing a new generation of axial fans, in addition to typical development targets such as efficiency and noise, other aspects must be considered. In evaporation applications for instance, icing of the fan blades and housing could lead to blockage of the motor and must be prohibited. Another important aspect in this application is the throw distance of the fan.

It does not seem to have that much potential to improve the mentioned development targets when further developing the state of the art concept of a fan stage of an axial impeller with tip gap and guide vanes which are often part of the fan housing. So, the decision is made during the development process to take a different approach: A shrouded impeller with a slightly diagonal approach combined with an inlet nozzle such as typically used in combination with centrifugal fans. The housing has guide vanes included and deflects the airflow into the axial direction to keep the outlet characteristics of an axial fan. During the development process, the structural mechanics and the aerodynamics of the fan are simultaneously optimized by using an automated workflow based on a parametric CAD model. This approach reduces the number of development loops and thereby the development time.

The Paper describes the tasks, the used tools and the results of the development process for the novel axial fan concept. As a result of the revised fan concept and the use of parametric optimization, the new fan has a higher efficiency than common axial fans, a steeper pressure characteristic with higher stall margin, a good noise behavior, less icing on the fan blades in evaporator applications and a higher throw distance. In addition to that, this fan has less increase of noise than typical axial fans, when it is used in applications with disturbed, highly turbulent inflow conditions.

Daniel.Gebert@de.ebmpapst.com

DESIGN AND OPTIMIZATION OF A DOUBLE-INTAKE AND ROTOR SQUIRREL CAGE FAN USING OPENFOAM AND METAMODELS

Alla Eddine BENCHIKH-LE HOCINE Université de Sherbrooke (Canada)
Sébastien PONCET Université de Sherbrooke (Canada)
Hachimi FELLOUAH Université de Sherbrooke (Canada)

The range hood is crucial in kitchens during cooking activities. Inside the residential houses, cooking activities are one of the main sources of particle emission, which decreases the air quality level. Furthermore, multiple studies found a strong correlation between the particles emitted from cooking activity and chronic obstructive pulmonary disease (COPD), lung cancer and diabetes. The use of an efficient range hood is essential to maintain a healthy air quality level inside the house. The fan is the main component inside a range hood. Most of the range hoods are equipped with an axial fan or a one rotor squirrel cage fan.

In the present study, a powerful double-intake and rotor squirrel cage fan is designed and optimized by using a developed optimization process loop based only on open source libraries. Dakota is used to achieve the sampling and build the surrogate surfaces, Salome to generate the geometry and the mesh grid and OpenFoam for the calculations. More than eleven design parameters are selected in the impellers, blades and volute regions. The two objective functions: total efficiency and the generated noise are improved by maximizing and minimizing their values, respectively. The Latin Hypercube Sampling (LHS) method is selected to achieve sampling over more than 363 design parameters set. In order to model the turbulent flow, a 3D incompressible simpleFoam solver is used and coupled to the Multiple Reference Frame (MRF) approach. The Kriging and the quadratic polynomial response surface are used to expand the design space and improve the objective functions. The total efficiency is improved by 12 % and the noise is reduced by 2 sones compared to the initial design. The Kriging Metamodel predicts with less than 2 % the total efficiency and 1 % the generated noise compared to the OpenFoam calculation. A large 3D coherent structure is observed in the volute region with a scattered turbulent region near the outlet. The optimal design is validated at the design point against the produced prototype, with an error of 2.8 % and 1.3 % on the total efficiency and generated noise, respectively.

alla.eddine.benchikh.lehocine@usherbrooke.ca

ON THE APPLICATION AND LIMITATIONS OF SOUND-ABSORBING MATERIALS FOR AXIAL FAN BLADES

Felix CZWIELONG FAU Erlangen-Nürnberg (Germany)
Benedikt BERCHTENBREITER FAU Erlangen-Nürnberg (Germany)
Christof OCKER Aalen University of Applied Sciences (Germany)
Thomas GEYER Brandenburg University of Technology Cottbus-Senftenberg (Germany)
Gabriel STOCKMEIER Former FAU-Erlangen-Nürnberg (Germany)
Markus MERKEL Aalen University of Applied Sciences (Germany)
Stefan BECKER FAU Erlangen-Nürnberg (Germany)

The design process of axial fans is based on increasing the aerodynamic efficiency and reducing the sound emissions. Low noise emissions are regarded as a quality feature and further are needed to fulfill increasing legal regulations. In order to achieve the lowest possible noise emission of the fan, a large number of blade modifications have already been investigated. For example, attempts have been made to reduce the turbulent ingestion noise of the fan with serrated, wavy or slotted leading edges. However, these modifications often reduce the aerodynamic efficiency. Therefore, the challenge is to find modifications of the fan blades, which induce low noise radiation at high efficiency. Since it was found that changes in the shape of the blade usually result in a reduced efficiency, the focus is on preserving the blade in its basic form. The reduction of the sound emission will be achieved by changing the materials in the region of the leading edge of the axial fan.

Based on previous works on stationary airfoils, this study investigates the impact of porous materials in terms of noise reduction of axial fans. Porous materials with different acoustic impedance and airflow resistivity were selected in order to understand the physical noise reduction mechanisms. The material properties are determined experimentally using a two-port test rig. The axial fans are tested in a standardized axial fan test rig with regard to their acoustic and aerodynamic behavior. Two inflow conditions with different turbulence intensity are considered.

In this study two different approaches are taken to identify the noise reduction mechanisms. First, the integration of sound-absorbing materials to increase the acoustic absorption. Second, the integration of a structured porosity, which should influence the momentum exchange in the flow. The first study shows that acoustic absorption is not the main cause of the noise reduction. On the one hand, this is due to the small surface area of the sound absorbers and, on the other hand, to the fact that the insertion of the absorbers generate a disturbed flow over the fan blade. To generate a quiet basic flow, a structured porosity was integrated in the second study. Here, a higher noise reduction could be observed for the axial fans. Further, it could be shown that a connection between the suction and the pressure side of the axial fan is indispensable for sound reduction and that structured porosities create higher noise reduction compared to sound-absorbing materials.

cz@ipat.fau.de

NOISE REDUCTION OF A CENTRIFUGAL PLENUM FAN WITH LEADING EDGE OR TRAILING EDGE SERRATIONS

Ignacio ZURBANO FERNÁNDEZ CETIAT (France)
Alain GUÉDEL CETIAT (France)
Mirela ROBITU CETIAT (France)
Michel ROGER Ecole Centrale de Lyon (France)

Leading-edge or trailing-edge serrations have been used with some success as a noise-reduction technique applied to fixed airfoils in wind-tunnels. More recently, this technique has also been used on axial fans. A previous investigation by the authors also assessed the application of sinusoidal blade leading-edge serrations on a plug fan. In this study, these previous results are re-evaluated and corrected. Furthermore, new research based on blade trailing-edge serrations is also presented.

RANS simulations on a non-serrated baseline impeller have been performed to define an assumed optimum geometry of leading-edge serrations to reduce the noise. This geometry has been used as a reference to manufacture three impeller prototypes with various geometries of serrations defined by their amplitude and wavelength. Three impellers with trailing-edge serrations were also designed, based on design criteria for fixed airfoils. All these impellers had an iron-shaped geometry, deemed to be better than the more prevalent sawtooth or sinusoidal shape.

All six prototypes have been tested in a reverberant room, where noise and air performance were measured simultaneously. For each impeller, six fan operating points were tested, and for each point, the sound levels in narrowband and one-third octave band were measured at the fan inlet and outlet. This allowed assessing the effectiveness of serrations at different operating points while checking their impact on the performance curve and fan efficiency. The results were then compared with the reference fan without serrations.

Leading-edge serrations have yielded a mitigated sound power reduction (up to 1 dBA reduction in the overall sound power level) and only for some configurations. One of the impellers slightly reduced the noise at all the operating points, another did it at some of them and the third one increased the noise at most of the operating points. Trailing-edge serrations, on the other hand, reduced broadband noise for all operating points and impeller prototypes (an overall sound pressure decrease between 1 and 5 dBA). Leading-edge serrations reduce noise for low to mid frequencies but increase it over 1 kHz, whereas trailing-edge serrations reduce noise along the whole spectrum.

ignacio.zurbano-fernandez@capgemini.com

ON THE FLUID MECHANICAL AND ACOUSTIC MECHANISMS OF SERRATED LEADING EDGES

Stefan BECKER FAU Erlangen-Nürnberg (Germany)
Jörg RIEDEL FAU Erlangen-Nürnberg (Germany)
Manfred KALTENBACHER TU Graz (Austria)
Stefan SCHODER TU Graz (Austria)
Felix CZWIELONG FAU Erlangen-Nürnberg (Germany)

In most applications, axial fans are operated in disturbed inflow conditions. This is the case, for example, in air conditioning systems when the axial fan is positioned downstream of a heat exchanger. But axial fans are also used in traffic and habitation applications for cooling and fresh air supply. Here, the turbomachines are operated with protection grids or filters, which in turn can generate turbulence on the suction side of the axial fan. All of these induced inflow turbulences cause the sound radiation from the fans to increase and the systems to be recognized as disturbing. Especially due to the current corna-panedemic this disturbing factor increases, due to the fact that air conditioning units are more often installed in indoor areas. In order to reduce the sound emissions of axial fans under disturbed inflow conditions, leading edge modifications of axial fans have been investigated in recent years. For example, it has been attempted to reduce the sound emissions from the fans by means of serrations, slits or porous materials, while at the same time avoiding any losses in the pressure build-up. The basic mechanisms of these sound reduction measures are not yet fully understood and many leading edge modifications are based on internal company experience. In order to understand the physical mechanisms of serrated leading edges, simulations of a flat-plate fan downstream of a turbulence grid were performed. The simulations were validated with experimental measurements. It was shown that the serrations form longitudinal vortices along the fan chord and that this leads to a stabilization of the flow over the blade. In addition, radial flows are reduced by this fluid pattern and backflow over the tip gap are reduced. The improved flow guidance over the blade surface reduces the pressure fluctuations on the blade surface, which in turn leads to a decrease in sound radiation. A transfer of the sound reduction measures from a flat-plate fan model to profiled fans showed that the found fluid mechanical mechanisms have only a smaller impact on profiled fans. This indicates that the possibilities for optimizing axial fans with leading edge modifications are limited and can only result in noise reductions for certain cases.

sb@ipat.fau.de

MARKET SURVEILLANCE OF ECO-DESIGN REGULATION - ASSISTANCE FROM THE FAN INDUSTRY

Geoff LOCKWOOD European Ventilation Industry Association (Belgium)

Market surveillance is expensive, cheating is easy and the environment is the loser. Environmental regulations do have a positive impact, but unfortunately there are organisations that want to by-pass the rules. The European fan industry understands the importance of regulations and the difficulty of enforcement authorities with limited resources. They have produced guidance and tools to assist in identifying possible infringements.

The European Commission regulation 327/2011 eco-design requirements for fans driven by motors with electric input power between 125 W and 500 kW has transformed the market with a European Ventilation Industry Association (EVIA) survey estimating 160 TWh of electrical energy saved, 75 Mt CO_2, between 2012 and 2020. But every non-complying fan diminishes this impressive figure.

The scope of the survey was for fans within the scope of 327/2011 with an electrical input power of <22 kW. The survey identified that 80 % of those are integrated in other product. Within Europe these integrated components are typically placed on the market before being used by other manufacturers in other products, such as ventilation Units. The Market Surveillance Authorities (MSA) has an opportunity to enforce prior to subsequent use. But many complete products are imported with fans already integrated, here is an opportunity to avoid the regulations and a more difficult challenge for MSA's.

Physically testing fans to determine the performance and efficiency can cost thousands of Euros, and for some very large products it is not practical. The European Commission's INTAS project, that included Industrial Fans, confirmed the difficulty of surveillance of such products and proposed some measures. One proposal is a check list to identify potential non-compliance.

EVIA has taken this proposal, expanded the idea to a tiered approach and published in a guidance document. The tiered approach starts from a simple check of the product rating plate up to full product testing, including a check-test tool to check published product data. It can be used for any fan, from the smallest, ones integrated in other product to the very large.

Non-compliance can occur from misunderstanding or deliberately avoiding the requirements. The guidance can be used by manufacturers to understand their obligations. MSA's can use it to identify potential non-compliance and focus their limited resources.

geoff.lockwood@uk.ebmpapst.com

FAN EFFICIENCY REGULATIONS IN THE UNITED STATES

Michel IVANOVICH AMCA International (United States)

U.S. regulation of energy efficiency for commercial and industrial fans (CIF), i.e., those analogous to fans covered by EC 327 Lot 6, is still incomplete after more than a decade since rulemaking efforts started. The regulatory effort can be divided into three eras defined by presidential administrations. In 2011, during the Obama Administration, the U.S. Department of Energy (USDOE) initiated a CIF rulemaking in June 2011 by publishing a preliminary determination that USDOE had the authority to regulate CIF. This effort culminated in 2015 with a "term sheet" publicly negotiated by USDOE and stakeholders representing manufacturers, advocacy organizations, and industry associations. The term sheet outlined a proposed structure of a USDOE fan-efficiency regulation, including scope, metrics, and types of fans to be included and excluded. The term sheet was important in that it solidified support for two new fan efficiency metrics, Fan Electrical Power (FEP) and Fan Energy Index (FEI). FEP and FEI are known as wire-to-air metrics because they consider the impacts of fans, motors, and drives toward the calculation of the metrics. Also, FEP and FEI can be calculated at any point on the fan-performance curve, enabling part-load efficiency determinations.

Progress on the fan regulation then stalled during the Trump Administration by executive order halting new USDOE regulations. Air Movement and Control Association (AMCA) International remained engaged in fan regulation during this time, first by completing two standards – AMCA 207, for determining part-load losses of certain motors and drives, and AMCA 208, for calculating the Fan Energy Index, and then by advocating for the adoption of the FEI metric in model energy codes that would ultimately be adopted into state energy codes. ASHRAE Standard 90.1-2019 and the 2021 International Energy Conservation Code adopted FEI, referencing ANSI/AMCA 208-18. Florida and Oregon became the first states to use FEI in state energy codes, followed by California for its 2022 commercial energy code. Also during this time, the California Energy Commission initiated development of a fan regulation under its state authority for appliance efficiency regulations, known as Title 20. Early work on the yet-incomplete regulation adopted much of the 2015 USDOE term sheet and the FEI metric.

Also during the Trump administration, AMCA integrated portions of AMCA 207 and 208 into an FEI “test procedure” that could be adopted by regulators, referencing ANSI/AMCA 210 and ISO 5801 as test methods for CIF. ANSI/AMCA 214 was published in January 2021, at the start of the Biden Administration, with the purpose “to aid federal and state rulemaking efforts to establish energy-efficiency standards for commercial and industrial fans and blowers, providing a consistent method of calculating FEI…”

In 2021, USDOE published a final determination that it had the authority to regulate CIF, signaling a re-start of the CIF rulemaking. In doing so, USDOE adopted the AMCA 214 definition of Fan, and, in a separate preliminary rule, expanded the scope of the CIF regulation to cover circulating fans that are not ceiling fans.

This presentation will outline how CIF are currently regulated in U.S. state energy codes and where a federal CIF regulation seems to be heading based on the 2015 term sheet and the draft California CIF regulation.

agunzner@amca.org

FANS ENERGY EFFICIENCY: THE LATEST REGULATORY DEVELOPMENTS IN THE EU.

Ronald PIERS DE RAVESCHOOT European Commission (Belgium)

The recent developments in Europe such as the war in Ukraine and associated rise in energy prices has made the energy transition more important and urgent than ever, in order to reduce the EU energy dependency and fight climate change. Energy efficiency remains among the top priority of the EU policy makers. The recent policy developments in the EU will be presented, focusing on those affecting the energy efficiency of energy-related products (Ecodesign and Energy Labelling Regulations). This will include the newly adopted Ecodesign and energy labelling Working Plan 2022-24, and the Proposal for a new 'Ecodesign for Sustainable Products' Regulation (ESPR), both from March 2022.

The most recent developments with respect to electric motors and variable speed drives will be briefly reminded (Regulation (EU) 2019/1789 which entered into force in July 2021) and then an update on the review of the fans (EU) 327/2011 Regulation will be provided, following the Consultation Forum held in April 2022.

ronald.piers-de-raveschoot@ec.europa.eu

TRACEABILITY OF TEST RIGS TO NATIONAL STANDARDS

Maxime LE RALLIC EP Ehrler Prüftechnik Engineering GmbH (Germany)
Kerstin BAUEREISS EP Ehrler Prüftechnik Engineering GmbH (Germany)
Leidig HEINZ EP Ehrler Prüftechnik Engineering GmbH (Germany)

Manufacturers of fans and ventilators use special fan test rigs during research and development and as end-of-line test benches for the production facilities. There are many kinds of constructions available, which are harmonized by standards like DIN EN 5801 and AMCA. Nevertheless, have you ever asked yourself the critical question: Can we believe the results? How valid are the measured values actually? Are the results of our different test equipment at different locations comparable?

The only thing that can give you security is a so-called transfer standard.

We provide an overview of the calibration pyramid: from the fan itself, over the working standard fan test rig, the transfer standards and primary standards up to the SI units of length and time. To ensure security about your data, complete traceability back to the SI units is essential. The transfer standard can be used by a calibration laboratory or by a manufacturer himself. The transfer standard is connected to the fan test rig and does not calibrate the fan, but the test rig. It closes the gap between working standard and primary standard in order to achieve full traceability.

We show all the criteria that must be met by a volume flow measurement system in order to work as a good transfer standard. In a concrete example project, we present a transfer standard covering a flow rate from 4 to 40,000 m^3/h, which serves as a basis for the worldwide comparison of test stands of a well-know fan manufacturer both for test stands in blowing and suction operation. In order to cover a measurement range of 1:10,000, we have combined different measurement sections in one flow measurement system. Depending on the flow range that it to be covered by the transfer standard, different types of measurement sections can be advantageous. We take an overview of the advantages and disadvantages of different measuring sections. In addition, we focus on the measurement accuracy, the calibration process of the transfer standard itself and the calibration process of the test equipment with the transfer standard, as well as the ease of use and mobility of the system.

m.lerallic@ep-e.com

Explicit vs Implicit Numerical Modelling of Air-cooled Condenser Fans

Adam VENTER Stellenbosch University (South Africa)
Michael OWEN Stellenbosch University (South Africa)
Jacques MUIYSER Howden (Netherlands)

The numerical analysis of the flow field about large-scale air-cooled condensers (ACCs), which typically find application in the thermoelectric power industry, represents an enduring challenge to engineering efforts. Numerical analyses of ACCs are complicated by the scale of the systems and the innate complex flow phenomena that transpire during their operation. By virtue of their scale, computational fluid dynamic (CFD) models of ACCs are typically constrained to utilizing some or other implicit formulation to represent the system's large array of axial flow fans. However, customary implicit axial flow fan model formulations are only able to provide a limited approximation of actual fan performance within the ACC's complex flow environment, especially under conditions characteristic of operation during windy periods. Accordingly, these implicit fan models limit our ability to derive quantitative information from the associated ACC simulations over all operating points of interest. Therefore, to contribute towards deriving new understandings that can facilitate the improvement of these implicit fan models (and ultimately ACC wind effect analyses), an explicit 3D fan model capable of delivering detailed insight into ACC fan behaviour mechanics has been developed.

This explicit model is analysed under low inlet flow rate conditions characteristic of those under which actuator-disk type implicit fan models suffer. The aerodynamic information determined by the explicit 3D model is compared to the airfoil characteristics used by a 2D-referenced and a 3D-corrected implicit actuator-disk fan model. The differences between the data sets are then highlighted to gauge the potential shortcomings of the implicit models, guide improvement efforts and advance current understanding of 3D effects in the context of low pressure-rise, low hub-to-tip ratio axial flow fan operation. The results of this paper show the important influence the velocity evaluation method has on the description of 3D effects and the results suggest that potential improvements to the 3D-corrected actuator-disk model may be found through a revised treatment of the near-hub augmentation and the scaling of the airfoil polar correction with flow rate.

18224865@sun.ac.za

A Model Fan to Test the Train Piston Effect at Grand Paris Express Metro

Elisa BÉRAUD Eiffage Énergie Systèmes (France)
Benoît HOUSEAUX Eiffage Énergie Systèmes (France)
Fabrice DUET Eiffage Énergie Systèmes (France)

The "Grand Paris Express" railway network is currently under construction. We know that piston effect created by train operated over 100 km/h (65 mph) through a succession of 1,000 - 2,000 m tunnels and stations will differ from the traditional Parisian metropolitan network. The piston effect created in front and behind the train will be reflected in the shafts positioned in the tunnels for the access of the emergency teams. The equipment in the shafts, especially the fans, will be subjected to strong variations in pressure. Therefore, they will be operated in areas that are not defined by the usual for fans.

In order to characterize a "Grand Paris Express" type machine, Eiffage carried out a test bench to test the behavior of a Ø1000 model fan that has the same aeraulical and mechanical characteristics of the 300 fans that will be installed in the service structures of this new network. The model fan has been first tested in the unexplored area of negative pressure and negative flowrate to define the behavior in the 2 other quadrants of the curve. It is now operated with successive negative and positive forced pressure on thousands of cycles to ensure the durability of the equipment on several years of operation.

elisa.beraud@eiffage.com

EXPERIMENTAL INVESTIGATION OF THE BLADE TIP TIMING UNCERTAINTIES

Zhicheng XIAO Shanghai Jiao Tong University (China)
Yiming MENG Shanghai Jiao Tong University (China)
Hua OUYANG Shanghai Jiao Tong University (China)

The uncertainties of blade tip displacement measurement using BTT are investigated through experiments. A novel test bench is established where the blade deformation can be measured simultaneously by laser displacement sensors and BTT, which enables the determination of the BTT measurement error. It is discovered that the rotational speed fluctuation is the primary factor that degrades the accuracy of BTT. The magnitude of error is dependent on the probe position and could be up to 0.5 mm even when the rotor is driven by a servo. Fortunately, it is also validated that using multiple keyphasors could effectively elevate the accuracy of BTT measurement.

optimusxzc@sjtu.edu.cn

UNIFORMITY INDEX AS A UNIVERSAL AIR-COOLED CONDENSER FAN PERFORMANCE METRIC

Simon MARINCOWITZ University of Stellenbosch (South Africa)
Michael OWEN University of Stellenbosch (South Africa)
Jacques MUIYSER Howden Netherlands BV (Netherlands)
Peter HOLKERS Howden Netherlands BV (Netherlands)

Air-cooled condensers (ACCs) use axial fans to force airflow through finned tube heat exchangers and facilitate the heat transfer required to condense steam in a Rankine cycle. Distortions in the fan inlet airflow reduce volumetric flow rate (volumetric performance) through the axial fans and increase the dynamic loading of the fan blades. Severe dynamic blade loading can result in fan blade and motor gearbox damage, unit trips and related reductions in annual performance. One of the main contributing factors to distorted inlet airflow is ambient wind.

Literature relating to ACC fans and wind effects includes numerical and experimental studies that tend to focus on either volumetric performance or dynamic blade loading due to the difficulties related to determining the former in experimental work and the latter in numerical work. That these performance metrics are related is implicit since they are both affected by the same phenomena (distorted inflow). However, due to the lack of comprehensive studies examining both metrics, the relative sensitivities of these metrics to wind effects are not understood.

This study used a numerical approach to simultaneously investigate ACC fan volumetric performance and dynamic blade loading under windy conditions. Based on the numerical results, a fan inlet airflow uniformity index was identified as a single metric that can be used to estimate the severity of both the reduction in fan volumetric performance and the increase in dynamic blade loading under windy conditions. A proportional relationship between uniformity index and both fan volumetric effectiveness and dynamic blade loading was observed. The same proportional relationship was observed in experimental data from on-site ACC testing. Thus, suggesting uniformity index can be used in numerical or experimental work to allow for a qualitative investigation of wind effects and wind effect mitigation mechanisms.

17509319@sun.ac.za

Sound Generation and Propagation in a System Consisting of Two Periodic Rows of Channels

Mohcene OULMI École Centrale de Lyon (France)
Michel ROGER École Centrale de Lyon (France)
Benali BOUALEM Alstom Transport (France)
Quentin TRAHARD Alstom Transport (France)

The aim of this work is to improve the design of the cooling fan system integrated in the traction motors of trains in order to reduce the aerodynamic noise. The latter is the dominant noise contribution at high rotation speeds. The use of numerical simulations to predict the noise remains very expensive, especially when several geometrical parameters must be tested. The analytical methods are better suited at the early design stage due to the very low computational time compared to numerical simulations. The methodology is to split the system into generic components addressed separately.

The present work is focussed on the system consisting of guide vanes and cooling channels. This system can be considered as two periodic rows of channels separated by a small distance.

The first mechanism investigated in this study is the sound generation by the impingement of the wakes of an upstream impeller with circumferentially unsymmetrical blade-spacing on the guide vanes. The second one is the transmission of the acoustic waves generated by the latter through a row of thick-walled channels. These waves are partially reflected by the channels, generating an upstream field. Back-and-forth acoustic waves develop between the channels and guide vanes. The sound propagation in this complex geometry is achieved by the use of two transmission models which take into account the wall thickness. A two-dimensional analytical approach is used, based on the mode-matching method. The modal expressions are written in each domain and matched according to conservation laws of fluid dynamics. An iterative method is used in order to take into account the multiple reflections of the acoustic waves between the guide vanes and the channels.

The influence of various parameters on sound generation and transmission, such as the frequency, the modulation angles of the impeller blades, the dimensions of the guide vanes, those of the cooling channels and the distance between the guide vanes and channels is discussed in this work.

mohcene.oulmi@ec-lyon.fr

ACOUSTICS OF A BACKWARD CURVED RADIAL FAN – CAA SIMULATION AND EXPERIMENTAL VALIDATION

Andreas LUCIUS ebm-papst Mulfingen GmbH & Co. KG (Germany)
Rebecca SCHÄFER Institute of Fluid Mechanics and Turbomachinery, TU Kaiserslautern (Germany)

The acoustic emission of radial fans is an important issue in product development. In addition to experiments in acoustic test rigs numerical methods represent alternatives to predict sound emission and to gain insight into sound generation mechanisms. In this project a backward curved radial fan is analysed in two configurations: fan alone and fan installed in spiral casing. Experimental investigations were done at the anechoic fan test rig at TU Kaiserslautern. The experimental data was used to validate CAA Simulations based on finite volume LES. Two methods were used for the calculation of sound at the microphone positions: the direct evaluation of the compressible pressure and far field pressure evaluated with the FW-H (Ffowcs Williams and Hawkings) equation. The FW-H integral equation assumes free-field radiation from the sound emitting surfaces. The method is applied for both configurations, although assumptions are not valid the fan in spiral casing. In contrast the direct method requires sufficient mesh resolution in the propagation zone in order to resolve the sound waves. Mesh size studies were done to investigate the required number of cells per wave length. A 3rd order scheme for discretization of the convective terms was applied, which resulted in very good agreement of measured acoustic spectra with 10 to 15 cells per wave length.

In case of the fan alone a periodic simulation model was applied in comparison to a full rotor simulation. Good agreement of FW-H and direct acoustics were observed. The limits of the periodic approach were investigated. The approach is not valid at operating points with large unsteady separations. In addition the periodic simulations result in increased levels due to the coherent sound field. On the axis of rotation a simple correction of the sound pressure level is applicable to obtain the physical uncorrelated sound emission. For pressure probes off-axis increased levels compared to full model simulations were observed. This is especially observed for low frequencies.

andreas.lucius@de.ebmpapst.com

AEROACOUSTIC SIMULATIONS OF AN AXIAL FAN WITH MODELLED TURBULENT INFLOW CONDITIONS

Philipp DIETRICH ebm-papst Mulfingen GmbH & Co. KG (Germany)
Marc SCHNEIDER ebm-papst Mulfingen GmbH & Co. KG (Germany)

Evaluating and reducing the emitted acoustics is an important part in the development of new generations of fans. Especially for fans, which operate in applications close to humans (e.g. air conditioning, kitchen hoods, etc.), low noise generation is requested. In such applications the acoustics may differ significantly compared to measurements or simulations of the standalone fan due to the turbulent character of the inflow conditions. For axial fans these turbulent inflow conditions are often caused by upstream heat exchangers. Due to the complex geometry and dimensions of typical heat exchangers, it is economically not possible to properly resolve the turbulence generating geometry in scale resolving aeroacoustic simulations (LES).

The FRPM (Fast Random Particle Mesh) tool, developed by the DLR Braunschweig is able to reconstruct turbulent fluctuations based on statistical turbulence properties (e.g. turbulence intensity and length scale) derived from RANS simulations. Using this method it is possible to synthesize the time dependent turbulent fluctuations caused by complex geometries, such as heat exchangers, using precursor RANS simulations. The reconstructed turbulence can then be used to model the inflow conditions due to the presence of a heat exchanger in scale resolving aeroacoustic simulation of fans. This approach allows the evaluation of the effect of turbulence generating geometric features on the acoustics of the fan with feasible computational effort.

The developed approach is tested by simulating a ducted axial fan, which was experimentally investigated at the University of Siegen. In the experiment the turbulent inflow conditions are generated by a turbulence grid. Besides the extensive experimental investigations, the setup of this experiment allows the simulation of the fan and the turbulence generating grid with reasonable computational effort. The simulation results with the modelled inflow turbulence show good agreement with the measurements as well as a simulation of the entire setup. However, due to the missing of the turbulence generating geometry as well as additional upstream geometries in the simulation with reconstructed turbulence, the effect of these features on the acoustic propagation cannot be represented.

philipp.dietrich@de.ebmpapst.com

ACCURAY AND ADVANTAGES OF HYBRID SOLUTIONS COMPARED TO DIRECT NOISE CALCULATIONS FOR LOW-SPEED FAN NOISE

Korcan KUCUKCOSKUN Siemens (Belgium)
Axel KIERKEGAARD Siemens (Germany)

In most industrial applications it is widely accepted that noise is a concern for fan manufacturers due to comfort trends and regulations. However, fan noise prediction is not a headache for engineers anymore thanks to advanced and mature simulation techniques and increasingly available computational resources. Based on their needs, engineers can still choose methods such as:

- a direct noise calculation from a compressible CFD solution as a reference solution

or hybrid solutions where;

- the compressible flow results are coupled with a time domain acoustic propagation technique
- the incompressible flow results are coupled with a time domain acoustic propagation technique
- the compressible flow results are coupled with a frequency domain acoustic propagation technique
- the incompressible flow results are coupled with a frequency domain acoustic propagation technique

and other non-listed methods.

This presentation focuses on the blade loading noise generated by a virtual low-speed five-blade axial fan which operates under a turbulent jet stream. All five alternative techniques listed above are applied to this fan using commercial finite volume software, which is coupled with a commercial finite element software for the last two bullet points above. It is worth to note that the frequency domain acoustic propagation solution uses a high order finite element model and reconstructs the source terms in frequency domain.

First, the competitiveness is challenged in a free-field radiation setup, to demonstrate that all five simulation approaches converge to the same acoustic levels in an anechoic-room like environment. Both tones and broadband responses are cross-validated using a finite-volume solution in one hand, and a finite element model on the other hand.

Secondly, the hybrid solution is applied to the free-field solution once more, in order to perform a contribution analysis between two zones representative of the leading-edge and trailing-edge of the blades. The leading-edge zone is found to be dominant in the simulations as expected since the fan operates downstream of a jet. This shows how a hybrid solution complements the workflow and how further acoustic insight can be obtained without rerunning the flow solution.

Finally, the acoustic response of an installed fan is computed using the same flow results around the axial fan. The reflection and scattering due to surrounding surfaces are then considered in the finite element acoustic solution. This shows how a frequency domain acoustic solution complements a time domain finite volume flow solution in presence of installation effects.

korcan.kucukcoskun@siemens.com

INVESTIGATION OF FORWARD AND BACKWARD SWEEP AXIAL FAN NOISE SOURCES USING LATTICE-BOLTZMANN METHOD

Dipali GHODAKE Université de Sherbrooke (Canada)
Marlène SANJOSÉ École de Technologie Supérieure (Canada)
Stéphane MOREAU Université de Sherbrooke (Canada)
Manuel HENNER Valeo Thermal Systems (France)

Low speed axial fan used in automotive industry contributes to rising noise pollution levels in the environment that is necessary to mitigate. In this study, we simulated the impact of sweep on noise levels and compared with unswept fan by using Lattice-Boltzmann method. Different inlet configurations study showed that in the experiment the strong inflow distortion is present. The quality of an incoming flow field is also responsible for higher broadband levels and tonal noise in absence of a stator in the spectra. The leading edge and trailing edge flow field details agrees very well with experimental recordings. The analysis showed that the swept blades alter the aerodynamic flow field. The backward sweep exhibit reduced blade loading near tip and increased leakage flow. The weak turbulent structures formed due to roll up of leakage flow that interacted with the blade near tip and acts as a major noise source. The standard deviation of pressure contours on blade suggest the potential noise sources. Finally, the noise spectra from simulation is validated with experiment showed a very good agreement.

dipali.ghodake@usherbrooke.ca

INVESTIGATIONS CONCERNING THE FLOW STABILIZATION OF BACKWARD CURVED CENTRIFUGAL IMPELLERS AT LOW FLOW RATE

Frieder LÖRCHER ZIEHL-ABEGG SE (Germany)
Sandra HUB ZIEHL-ABEGG SE (Germany)
Marlène SANJOSÉ École de Technologie Supérieure (Canada)
Stéphane MOREAU Université de Sherbrooke (Canada)

At part load (at lower flow rate than the design flow rate), the flow pattern of backward curved, housingless centrifugal fans gets more and more unstable (unsteady) with sinking flow rate. In direct consequence, both a significant increase in noise emission, appearing especially as distinct subharmonic humps in the sound power spectra, and a decrease of the fan efficiency can be observed. By means of unsteady CFD simulations, which are validated by experimental results, the mechanisms for this undesired behavior have been investigated. It turns out that the phenomenon is self-induced by the impeller and does neither correlate to any upstream turbulence or flow inhomogenouities, nor any influence of the flow in the leakage gap can be identified. Instead, it seems to be induced by flow structures located on the downstream side of the impeller, where strong vortical structures, rotating in circumferential direction at a rotation speed with slip to the impeller, can be observed. These vortical structures are fed by detaching vorticity of highly loaded impeller blades, and they grow and change their structure with sinking flow rate. It was found out, that by controlling and partly avoiding these pressure-side rotating flow inhomogenouities, the part load behavior of the fan can be significantly improved at low flow rates. The result is that the impeller efficiency and the pressure rise can be enhanced at the corresponding duty points, whereas the noise generation can be strongly reduced. Thus, the operating range of the fan can be significantly extended towards lower flow rates. The numerical results are confirmed by some experimental results.

frieder.loercher@ziehl-abegg.de

INVESTIGATION OF VORTEX-GENERATOR INDUCED FLOW STRUCTURES ON A FLAT PLATE USING IR THERMOGRAPHY, PARTICLE IMAGE VELOCIMETRY AND OIL FLOW VISUALIZATION

Alexander SCHUKMANN Technical University of Kaiserslautern (Germany)
Christian LANDFESTER Technical University of Kaiserslautern (Germany)
Martin BÖHLE Technical University of Kaiserslautern (Germany)

The use of passive flow control, e.g. by means of vortex generators attached to the blade or endwall surface, is one approach to reduce boundary layer separation induced aerodynamic losses in axial fans by separation point delay. An efficient use of this approach requires a sufficient analysis and understanding of the three-dimensional flow structures.

Even though computational fluid dynamics (CFD) methods based on the Reynolds-Averaged-Navier-Stokes equation approach (RANS) occupy a significant position in the development phase of aforementioned machines, experimental validation of novel, but also established numerical models is still indispensable and essential. Especially concerning flow separation, CFD models inhabit an inevitable sensitive dependency of separation point prediction on mesh quality and mesh characteristics.

In this work we present a detailed and comprehensive study of the flow structures on a flat plate using several established vortex generator geometries, i.e. spheres, cones, ramps and wheeler-type wedges. The experiments are conducted in a plane wind tunnel with Reynolds numbers about $Re \approx 1e05$ at a fixed turbulence intensity of 1.5 % and under steady-state flow conditions. The surface flow structure is investigated using oil flow visualization and IR thermography utilizing the dependency of wall heat transfer on local flow characteristics. For this purpose, the flat plate is heated internally through a temperature-controlled adhesive thermo polyester heating foil. The plate surface was coated with a thin, low heat conducting polymer (PTFE) limiting the heat flux and allowing local flow phenomena to be resolved sufficiently. For a comprehensive insight into the three-dimensional flow field, Particle Image Velocimetry (PIV) measurements were conducted on several planes parallel and perpendicular to the main flow direction, supplementing the above-mentioned surface flow analysis.

This investigation provides a broad description of local three-dimensional flow phenomena induced by common vortex generator designs, applicable to boundary layer separation control in axial fans. Within the scope of this study, it is shown that IR thermography is a suitable approach for contact-free measurements of surface flow structures.

alexander.schukmann@mv.uni-kl.de

NUMERICAL INVESTIGATION OF AN AXIAL FAN FOR AUTOMOTIVE APPLICATIONS

Nicola ALDI University of Ferrara (Italy)
Nicola CASARI University of Ferrara (Italy)
Stefano OLIANI University of Ferrara (Italy)
Michele PINELLI University of Ferrara (Italy)
Enrico MOLLICA SPAL automotive (Italy)
Filippo MENICHINI SPAL automotive (Italy)

Automobiles and motorbikes need cooling circuits to remove heat from their engines. Heat is rejected to the coolant that transfers it to the surrounding environment in the radiator. In order to maximize the heat transfer at the radiator, fans are employed. These fans are typically of the axial flow type, given the requirements of high flow rate with relatively low pressure drops.

The requirements of high efficiency and low noise of axial flow fans are pushing manufacturers and researchers to look into the operation of these machines in the effort of minimizing losses and noise source. For this reason, ad-hoc test benches are developed for the experimental analysis. Over the last decade, numerical simulations with CFD software have become an industrial standard for complementing and getting insight into the experimental evidence and propose solutions. Among the possible CFD suite available, OpenFOAM is gaining an increase share of the market due to its opensource nature and the great accuracy and reliability of the results.

In this work the authors presents the CFD investigation of an axial flow fan with OpenFOAM. The results are compared with experimental data obtained in twoa test benches: one ad hoc developed for acoustic and flow field analyses and one build in compliance with ANSI/AMCA Standards for performance determination. The capability of the software of correctly replicating the fan performance and flow filed even in this complex set-up (due to the loose constraints of the fluid flow) is shown both with steady and transient simulations.

nicola.aldi@unife.it

INTRODUCING A HIGHLY EFFICIENT CFD SOLVER FOR LOW-SPEED FAN ANALYSIS

Domenico MENDICINO Cadence Design Systems (Belgium)
Wout PONCELET Cadence Design Systems (Belgium)

Running a numerical CFD simulation on a low-speed fan is found to be highly challenging as the flow is of low-Mach and generally consists of separations and complex flow structures. The robust, cost-efficient, and accurate pressure-based solver proposed by Cadence Design Systems, allows one to have a good understanding of low-speed fan performances and characteristics which is of critical importance in most working environments.

Typical density-based CFD solvers, solving the Navier-Stokes equations for continuity, momentum, and energy, are highly efficient to solve high-speed flows, but generally show a slow convergence speed for low-speed applications without dedicated preconditioning techniques. The simulation of low-speed flow fans can be accelerated through a pressure-based solver which directly solves for the pressure and momentum. The integration of the coupled pressure-based solver into Cadence's Fidelity environment, including best practices and interfacing, is presented. The solver is built on a fully coupled implicit all-Mach methodology.

The coupled pressure-based solver is used to compute the performances of a squirrel cage-type fan: commonly utilized for heating, ventilation, and air-conditioning applications. The fan consists of a centrifugal impeller and a volute: the impeller is typically a low-speed forward-curved centrifugal wheel, the volute tends to be a simple scroll with a rectangular cross-section. The coupled pressure-based flow solver is used to simulate and analyze the complex flow field, more specifically the strong impeller-volute interaction near the volute tongue and the flow separation due to rapid flow turning. The impact of different turbulence models is examined and discussed. The time-dependent nature of the flow is also verified through an unsteady simulation of the flow field. A variable time step is considered throughout the simulation to decrease computational time and save engineering resources. The results are compared to the ones of steady RANS computations.

domenic@cadence.com

DESIGN OF A SMALL AXIAL-FLOW FAN WITH 0.2 HUB-TO-TIP RATIO

Massimo MASI University of Padova (Italy)
Andrea LAZZARETTO University of Padova (Italy)
Piero DANIELI University of Padova (Italy)
Carlo BETTANINI University of Padova (Italy)

This paper presents the arbitrary vortex design of a 315 mm industrial fan prototype obtained by using a practical design approach recently proposed by the authors. Aim of the paper is to explore the capability of the suggested design method in a 0.2 hub-to-tip ratio application. This very low hub-to-tip ratio has been chosen because design examples with such a small hub-to-tip are rather rare in the literature. In fact, common design approaches relying on the cylindrical surfaces flow approximation are conceptually not suited to machines which design point operation features a strong flow recirculation in the innermost region of the blade span and a marked radial shift of the meridional flow in the remaining part of the blade span. Although there are examples of industrial fans with hub-to-tip ratio even lower that 0.2, the value of 0.2 should be considered as practical effective lower bound. In fact, existing applications with hub-to-tip ratio lower than 0.2 are usually coupled with hub disk solution and with a non-aerofoil inner part of the blades. Thus, it is expected that the aerodynamic performance of these fans is almost not dependent on the design of the blade portion close to the hub zone.

In the paper, the presentation of the new 0.2 fan prototype is followed by the report of the fan aerodynamic performance, measured on an ISO 5801 test rig. These data are compared with the corresponding data of a 0.14 hub-to-tip ratio which incorporates aerofoil blades only in outer part of the span (radial coordinate - made dimensionless with the hub radius - higher than 0.2). Since the data available for the 0.14 hub-to-tip ratio have been measured for a 1845 mm industrial fan, a 315 mm scaled version, in which the hub has been enlarged to incorporate the non-aerofoil part of the blade, has been printed by rapid prototyping to estimate the decrease of aerodynamic performance due to the smaller size of the prototypes and to predict the aerodynamic performance achievable from the design obtained using the design approach proposed by the authors whether the machine is manufactured in large size. The results demonstrate the effectiveness of the design approach and quantify the combined effect of size and Reynolds number on the fan performance.

massimo.masi@unipd.it

DEVELOPMENT OF PRESSURE REGAIN UNITS TO INCREASE EFFICIENCY OF FREE-RUNNING RADIAL FANS

Philipp OSTMANN RWTH Aachen University (Germany)
Dimitrios PETROU RWTH Aachen University (Germany)
Martin KREMER RWTH Aachen University (Germany)
Dirk MÜLLER RWTH Aachen University (Germany)

To meet the rising comfort requirements of non-residential buildings like office buildings a mechanical ventilation in the form of e.g. air handling units (AHU) is needed. As of today, fans in AHUs are usually mounted in a free-running configuration, meaning that almost all kinetic energy present at the exit of the fan impeller is treated as a loss. This is due to the fact, that only the total-static pressure rise is taken into account to rate the fan performance. Depending on the operating point the ratio of dynamic and static pressure rise at impeller outlet can be unfavourable, leading to suboptimal efficiency of the whole system. This leads to higher electrical consumption than needed and therefore higher CO_2 emissions.

The present work investigates the behaviour of a generic radial fan in a free-running configuration by conducting 3D numerical flow simulations. In a first configuration, the radial fan blows into an open air duct channel. The kinetic losses as well as the static and total pressures at the impeller outlet are then evaluated over the operating range of the fan. In total three lines of constant fan speeds with eight operating points per speed line are investigated. By defining areas of occurring backflow downstream the fan, the areas of disadvantageous flow are identified.

To enhance the conversion of dynamic into static pressure and therefore increase the total-static efficiency an additional so-called pressure regain unit-body (PRU) is implemented in the flow downstream of the fan. The design is kept as simple as possible to enable an easier implementation in a real application. Starting from an initial design of the PRU the design is further improved to reveal possible limits of the efficiency increase by such unit. By designing the PRU in a specific way, it might be possible to shift the peak efficiency of the system towards lower or higher volume flows and thus better tune the system to the requirements.

philipp.ostmann@eonerc.rwth-aachen.de

AXIAL IMPELLER-ONLY FANS WITH OPTIMAL HUB-TO-TIP RATIO AND ADAPTED BLADES

Thomas CAROLUS Steinbeis-Transferzentrum FLOWTRANS (Germany)
Konrad BAMBERGER Private (Germany)

This study targets at determining impeller-only axial fans with optimal hub-to-tip ratio for highest achievable total-to-static efficiency. Differently from other studies, a holistic approach is chosen. Firstly, the complete class of this fans is considered. Secondly, the radial distribution of blade sweep angle, stagger angle, chord length and blade camber are varied to adapt the blades to the complex flow in the hub- and tip region. The tool being used is an optimization scheme with three key components:

(i) a database created beforehand by Reynolds-Averaged Navier-Stockes (RANS-) predicted performance characteristics of 14,000 designs,
(ii) an artificial neural network as a metamodel for the fan performance as a function of 26 geometrical parameters.,
(iii) an evolutionary algorithm for optimization, performed with the metamodel.

In general, the hub-to-tip ratios for the class of axial impeller-only fans proposed by the optimization scheme are smaller than those obtained applying the classic rules. A second outcome are the shapes of adapted blades which deviate substantially from the classic and even the state-of-the-art "swept-only" or "swept with dihydral" design. Chord length, stagger and sweep angle are distributed from hub to tip in a complex manner. The inherent reason is that the scheme tries to minimize not only the dynamic exit loss but also frictional losses due to secondary flows in the hub and tip region which eventually results in the maximum achievable total-to-static efficiency.

thomas.carolus@stw.de

CFD Analysis of Unsteady Flow in Non-uniformly Distributed Fan Blades

Tayeb AIOUAZ CETIM (France)
Samir ASSAF CETIM (France)

Advanced aerodynamic and aeroacoustic multiphysics modeling and simulation are becoming a major topic in the modern aerospace and automobile industry and even more in the design of HVAC (Heating, Ventilation and Air-conditioning) and cooling fan industries. Acoustic noise generated by a flow can be created through different mechanics but is ultimately due to fluctuations in the flow. The aeroacoustic computation of airborne noise requires highly accurate numerical approaches to deal with the complexity of phenomena involved, such as turbulent flow over solid bodies, high-speed turbulent shear flows, structural vibration that is induced by fluid flow, turbulent combustion, and laminar instabilities. Moreover, the requirements in terms of time and space resolution for the aeroacoustics as well as the identification and calculation of aeroacoustic sources make the computation of aerodynamic generated noise often extremely time demanding.

The objective of the present study is to reduce tonal noise of fans with non-uniformly distributed fan blades and to compare results from a light and fast Ffowcs Williams-Hawkings (FW-H) Steady numerical solver to experimental results . The present paper describes Computational Fluid Dynamics (CFD) calculations to study complex phenomena related to flow and aeroacoustic noise in non-uniform fan blades. The studied phenomena are interactions, flows and perturbations induced by the rotating blades motion and their impact on the aeroacoustic behavior of the fan. Numerical simulation has been carried out using siemens STARCCM+ software. Investigation of the flow variables provided by CFD calculations were used as inputs in Ffowcs Williams-Hawkings equation to calculate sound pressure levels for related frequencies. Appropriate boundary conditions have been applied to ensure non-reflection of acoustic waves, and, in certain regions, to provide inflow and outflow of the aerodynamic field. At the end of work, numerical simulation is compared to experimental measurements and relationship between surface pressure fluctuations and the field noise signals is highlighted.

tayeb.aiouaz@cetim.fr

THE DEVELOPMENT OF A PREDICTION MODEL FOR THE ACOUSTIC QUALITY OF FANS

Stephan TÖPKEN Carl von Ossietzky University Oldenburg (Germany)
Eike CLAASSEN Carl von Ossietzky University Oldenburg (Germany)
Steven VAN DE PAR Carl von Ossietzky University Oldenburg (Germany)

Fan sounds are a typical part of environmental noise that humans hear in different everyday situations and they can play a major role in the appreciation of a certain product. Especially loud and unpleasant sounds can be detrimental in certain applications. Although it is well known that many technical acoustic measures like a-weighted sound pressure levels have limitations in depicting the perception of sound, they are still widely used because they are easily communicated and commonly used by engineers. However, using only a level-based measure as a sole indicator can be misleading in an optimization process especially for sounds differing in spectral content or temporal signatures. In order to enable a successful development of more pleasant fan sounds, it is therefore necessary to understand and characterize the perceptually relevant aspects of fan sounds and their impact on the evaluation of the sound in a certain context.

A model for the prediction of the acoustic quality of fans is currently under development in a project funded by the German research association for air and drying technologies (FLT e.V.). Based on extensive listening tests and a factor analysis, the most important perceived sound characteristics could be identified for a broad variety of different fan sounds. Utilizing the output of a standardized loudness model, two indicators were developed that characterize the most important spectral aspects and distinguish three major groups of fan sounds. The pleasantness of the fan test sounds was assessed with an indirect listening test method adjusting the level of a fan sound to make it equally preferred as a reference sound. The level differences between test and reference sound can then be interpreted as a level penalty or level adjustment for the fan sound.

The developed prediction model links the perceived sound characteristics described by the indicators with the fan sound evaluations from the listening tests measured as level differences. Using the model, differences in sound character can be translated to a dB value, which might be more commonly understood and more easily interpreted than a rating on a categorical scale.

At present, the model is extended to cover a range of absolute sound pressure levels between 45 and 75 dB(A) and to characterize the impact of tonal components on the (un-)pleasantness assessment of the fan sound more precisely.

stephan.toepken@uol.de

ACOUSTIC PERFORMANCES OF IRREGULAR BLADE SPACING FANS TESTED ON AN EXCAVATOR

Laurent BOLLADE ` DYVA (France)
Xavier CARNIEL Cetim (France)
Bastien BERNIER Cetim (France)

On mobile machinery, fans are often the dominant source of external noise, especially with the reduction in noise generated by internal combustion engines. The gradual arrival of electrified machines will not necessarily solve the problem, as the need for cooling the combustion engine will be transferred to the batteries or accessories, and fan noise is likely to emerge even more with a quieter engine.

Initial tests have shown that irregular fans have little impact on the overall A-weighted level but do reduce the level of blade passing frequency components. Manufacturers in the sector have expressed a desire to know the acoustic performance of fans with uneven blade spacing blades on a machine, among other noise sources.

For the purposes of the demonstration, 4 excavators were assessed and the one radiating most noise on blade passing frequencies was selected. Two irregular fans with different angular distributions were tested and compared with standard fan noise.

The paper presents the results of experiments carried out on these machines. The tests were done in a configuration close to that of ISO 3744 (sound power measurement).

The first step was to evaluate the contribution of the fan to the total noise of the machine. The result depends of course on the machine, but the contribution is in all cases important, even very important as regards the external noise, it is negligible for the noise in the cabin. The investigations were continued on a machine for which the fan noise has strong tonal components. Two Irregular Blade Spacing Fans were then designed using simple modelling to be compared with the original fan.

Measurements were made with the three fans in identical configurations at different engine and fan speeds. They confirm a first result previously acquired on a test bench, which is the conservation of the overall A-weighted level. The 1/3 octave spectra are also slightly modified.

The impact is important on the discrete frequencies for which we observe very different distributions of the acoustic energy between the characteristic frequencies of the engine and the fans. The influence on 3 psychoacoustic criteria is also examined.

In parallel, the influence of the Flexible Blade Extension which modify the shape of the spectra (decrease in broadband noise but increase in discrete frequencies) is analysed.

Perspectives are then provided for further investigation of this subject, both in terms of experimental and simulation aspects.

laurent.bollade@dyva.fr

THE MOTOR SYSTEMS TOOL - LATEST DEVELOPMENTS

Sandie B. NIELSEN Danish Technological Institute (Denmark)

One of the main goals of the 4E EMSA project is to upgrade the capacity of motor system users all over the world, by utilizing tools and guides etc. making everything easy accessible as online content. One approach to achieve this goal is the Motor Systems Tool.

The Motor Systems Tool, which was developed with full support from Danish public means, did at the launch in 2011 present a new, impartial calculation tool in which the efficiency of complete motor systems is calculated. The aim was to create a tool which can be easily handled, gives good technical support for choosing the correct combination of elements when creating the optimal motor system and is available for a broad audience.

Since the latest presentation at EEMODS ’17 development of the tool has taken place, and with the resent start of a parallel Danish project, even more development has been secured looking forward to the coming Fan2022 conference in Senlis. The focus of the coming developments will be a more integrated evaluation in terms of Eco-design requirements for both pumps and fans. The tool shall also be updated to match the upcoming extension of the scope of the motor regulation, new motor technologies, elaborated application calculation, energy evaluation and finally an improvement of the usability of the tool. The Tool also have focus on future integration of the latest developments within the international standardization of power drive systems (IEC 61800-9 series) as well as an integration of AMCA loss models for motors & drives.

The presentation will show and demonstrate the latest developments of the Motor Systems Tool with a few calculation examples combined with discussions of the models used.

sbn@dti.dk

TESTING VERIFICATION OF MOTOR SYSTEM EFFICIENCY MODELS

Tim MATHSON AMCA International (United States)

A driven fan consists of a traditional fan product combined with a motor and can also include a motor speed controller, as well as a transmission. Due to the immense number of possible permutations, it is normally impractical to directly measure wire-to-air fan performance at all possible operating points. Since individual power and transmission components are frequently manufactured by different parties, they are tested separately, if at all. To help this situation, testing and modeling methods have been developed to establish motor system performance at partial load conditions. Two of these methods are IEC 61800-9 and ISO 12759-6 (based on AMCA 207). The approach for each of these methods is distinct and will be discussed, along with a comparison to actual laboratory test results on AC induction motors and variable frequency drives.

tmathson@amca.org

BENEFITS THROUGH COORDINATION & ALIGNMENT OF IEC & ISO STANDARDS FOR ELECTRIC MOTOR DRIVEN SYSTEMS

Maarten VAN WERKHOVEN TPA advisors (Netherlands)

In the past 3 years representatives from industry, research and other stakeholders have made concrete steps into an intensified coordination of energy efficiency standards. This has led to the establishment of the ISO & IEC Joint Advisory Body JAG 22 (see presentation by JAG22 repr.). Founding TC's are ISO TC 117 (fans), and IEC TC 2 (motors) and IEC TC 22/SC22G (PDS).

The main focus lies into facilitating the exchange and coordination between ISO and IEC in the field of all types of Electric Driven Machine Units (EDMU); and identifying the relevant coordination issues and proposed solutions and describe these considerations or results of such exchange and coordination discussions for guidance, reference.

Working on improved global energy efficiency standards can bring benefits to all stakeholders involved. With the overall goal to reduce energy consumption and limit the amount of carbon entering the atmosphere. Where a regulator focusses on regulation that is effective, delivering savings and keeping a level playing field in place, the industry prime focus will be on delivering the best solutions to the market at competitive prices and service levels.

A key in the supply chain, i.e. from manufacturer to customer/end user, with the regulator defining an important part of the playing field, is the quality of data used: in the manufacturers catalogues, in the customer's and system integrator's procurement specifications and in the specific applicable regulations. With the qualitative high-level data appropriate tools can be developed as well, for system design and for building, supporting (inter)national policy and climate program development.

This becomes more acute through the globally identified need for more stringent measures for reducing the carbon entering the atmosphere. Concrete international initiatives work on the development of programs for increased product efficiency, expanding the scope of regulation from product to systems, and expanding the actual MEPS in place to a greater portion of the world. Any assistance through available aligned standards could help stakeholders in optimizing the potential portfolio of standards and regulations for energy efficient products, units and systems.

mvanwerkhoven@tpabv.nl

DESIGN AND INVESTIGATION OF A CONTRA-ROTATING CENTRIFUGAL FAN

Christian FRIEBE Institut für Luft- und Kältetechnik gGmbH (Germany)
Oliver VELDE CFturbo GmbH (Germany)
Karsten HACKESCHMIDT Institut für Luft- und Kältetechnik gGmbH (Germany)

Centrifugal fans are common in many fields of application. A centrifugal fan consists of a drive, an impeller, and a volute. The volute casing is designed to turn the dynamic pressure at the outlet of the fan impeller into static pressure and to guide the flow towards a connected geometry, e.g. a flange. However, centrifugal fans without a volute are being used more and more often. A common application is the installation in an air conditioning unit, where typically the dynamic pressure at the outlet is dissipated. In this way, both manufacturing costs and installation space can be saved. On the other hand it must be noted that the air at the outlet of the fan is strongly swirled, which may lead to inadequate acoustic and aerodynamic behaviour of the flow in subsequent devices. These two disadvantages of a fan without casing led to the consideration of a contra-rotating fan.

The opportunities of a contra-rotating fans are well known and examined for axial fans, but rarely investigated for centrifugal fans. In this approach we present the design and the measurement of a contra-rotating radial fan comprising two concentrically rotating impellers and two drives. The fan is designed for a volume flow of 150 m^3/h and 125 Pa (static pressure). The diameter is 150 mm and the rotational speed of the inner impeller (impeller A) and the outer impeller (impeller B) is 3000 min^{-1} and 1500 min^{-1}, respectively.

The impeller were designed by means of CFturbo latest approach for the calculation of contra-rotating fans. This design's prototype was created by means of 3d printing and measured on a test-rig as well as examined by means of CFD.

The results indicate that the simulations are in good agreement with the experiment and provide more detailed insights into the internal flow structure of the impellers. Based on these findings, investigations were made at the leakage point between the two impellers. The results of the test rig measurements and the laser optical measurements demonstrate the influence of different sealing geometries on the leakage.

It has been shown that, compared to a centrifugal fan without volute, a contra-rotating fan can significantly reduce the power loss due to swirl at the outlet and increases the power transfer to the fluid compared to a diffuser with guide vanes at the same time.

christian.friebe@ilkdresden.de

A Fully Reversible Axial Fan with Two Counterrotating Impellers

Konrad BAMBERGER Private (Germany)
Maximilian KLENK University of Siegen (Germany)
Thomas CAROLUS Steinbeis-Transferzentrum FLOWTRANS (Germany)

Some technical applications require a fully reversible fan. That means that the fan performance characteristics must be identical for both flow directions. Several design principles are known for both, centrifugal and axial fans. The objective of this paper is the design, manufacture and test of a small fully reversible axial fan.

The design chosen consists of two identical contra-rotating impellers without guide vanes, placed in mirror image to each other in a duct-type housing. The blades of the impellers are made of thin non-cambered airfoil elements of constant thickness, fully symmetric to the stagger line. Stagger angle and solidity of the blade cascade as function of the radius are optimized for maximum performance. Each impeller is operated by a separate motor with individual continuously variable speed. Of interest is the field of characteristic performance curves as a function of the impeller speed ratio.

Two 80 mm diameter impellers are designed and manufactured via 3D printing. Impellers and the electronically commutated motors are mounted in a precision aluminum tube via three circular struts.

The volume flow rate/total-to static pressure rise and efficiency characteristics are measured for several speed ratios. As targeted, one distinct speed ratio provides an exit flow without swirl, i.e. fan performance with maximum total-to-static efficiency. As an outlook several of these contra-rotating impeller assemblies could be placed in one duct without guide vanes in order to increase the pressure rise of the unit while maintaining full reversibility.

konrad.bamberger@web.de

EXHAUST AIR CONTROL CONE FOR OUTLET SPEEDS OF AT LEAST 7 M/S

Hermann MAUCH Colasit AG (Switzerland)

According to the guidelines VDI-2280:2005 "*Discharge conditions for organic solvents*" and VDI-3781:2017 Part 4 "*Environmental meteorology, discharge conditions for exhaust gases ...*", an exit velocity of at least 7 m/s vertically upwards should be aimed for a better distribution of the corrosive exhaust air, this ensures sufficient dilution and undisturbed removal of the exhaust gases with the free air flow.

The corrosion-resistant speed regulator developed for this purpose is an automatic, purely mechanical regulator for flow speeds that are constantly above a specified minimum speed. The almost maintenance-free speed controller works without external energy. Its smooth-running, centrally mounted inner cone made of polypropylene sets the desired minimum speed at the round gap outlet using minimal aerodynamic forces (pressure and friction) and the spring force.

Initially, purely analytical calculation approaches for the balance of forces between the weight force in relation to the compression spring force and resistance force are shown. Then, using CFD analysis with OpenFoam, the flow in the vertical pipe, the accelerated flow situation between the outer cone and the inner cone and the ejection flow at the outlet are examined. The graphic evaluation shows velocity fields at minimum and maximum volume flow, the pressure fields and streamlines. In addition, an unsteady flow situation is calculated in order to determine the free floating of the inner cone with the resulting height.

By blowing out at a minimum speed of 7 m/s vertically upwards over the roof of the industrial building, the exhaust air regulating cones dilute and discharge any hazardous substances that may be released in the laboratory in order to avoid health hazards via the breathing air. The discharge of laboratory exhaust air is described in DIN 1946 Part 7. Due to the aerodynamically truncated inner cone, there is no need to set an increase in the fan volume flow for lower exhaust air flows. This avoids increased pressure losses and greater drive energy expenditure. In addition, the sound power values of the exhaust air towers are reduced by the insulating inner regulating cone. The rainwater that penetrates less due to the blockage is discharged to the outside via a built-in condensate loop. This technically necessary component is not taken into account in the calculation. The optional side injection into the vertical standpipe is also ignored. Purely stationary considerations are given.

h.mauch@colasit.ch

THE NEW METHOD OF REGULATION OF CENTRIFUGAL FAN

Piotr ODYJAS Wroclaw University of Science and Technology (Poland)
Przemyslaw MOCZKO Wroclaw University of Science and Technology (Poland)

The paper will present new concept of centrifugal fan regulation with use of variable blade length system. Higher demands regarding low energy consuming devices do not omit fan what is underlined by internal regulations (e.g. EU Regulation 1253/2014). Regulation of fan is required in each system when constant value of flow and/or pressure cannot be ensured. Application of frequency inventers seems to be the most efficient method of regulation in case of centrifugal fan but has also some drawbacks. It requires additional power supply infrastructure. If fan is larger, costs of whole system increases. In case of large fans this costs are very high and change of rotational seed causes problem, since high inertia forces due to large rotating mass exist. Also, what is sometimes forgotten, costs of reactive power in such cases are very important from economical point of view.

Alternatively, especially in case of larger centrifugal fans, implementation of fan with variable geometry can be solution ensuring operation in wider range with the highest possible efficiency. The concept described in the work is based on fan with variable blade length. Change of blade length causes change of impeller diameter and as a result flow characteristic of fan. In the article analytical description of such fan will be presented. What is more result of tests on real object with new solution will be presented to show fan performance curves within whole range of regulation.

As showed currently done analytical calculations and CFD calculations, such solution can be an alternative to centrifugal fan with rotating end part of blades as well. Even if many difficulties exist in case of application of such system, this solution does not require additional costly power supply components and seems to be reasonable for big fans.

piotr.odyjas@pwr.edu.pl

IN-DUCT MEASUREMENTS OF FAN-NOISE IN AN ACOUSTIC FAN TEST RIG: ANALYSIS SUPPORTED BY THEORETICAL SOURCE MODELLING

Lukas KLÄEHN German Aerospace Center (DLR), Institute of Propulsion Technology (Germany)
Antoine MOREAU German Aerospace Center (DLR), Institute of Propulsion Technology (Germany)
Luciano CALDAS German Aerospace Center (DLR), Institute of Propulsion Technology (Germany)
Ulf TAPKEN German Aerospace Center (DLR), Institute of Propulsion Technology (Germany)

A comprehensive experimental study is conducted at a new aeroacoustic fan test rig with the objective to improve the understanding of the dominant fan noise source mechanisms. For this purpose, the aerodynamic properties are resolved with hotwire probes and total pressure rakes and the acoustic characteristics are resolved with arrays of wall-flushed microphones in the inlet section. The interpretation of the results is supported by a physics-based, analytical fan model. The presented study focuses on the rotor-incoherent noise components. They are obtained by a cyclostationary analysis, which ideally removes tonal components from the sound pressure spectrum. The remaining spectrum consists of broadband noise and narrowband spectral components. The narrowband components are located around the blade passing frequencies and are associated with elongated turbulent structures, which create partly correlated noise sources when interacting with the rotor. The sound power of the broadband and narrowband components is determined with help of a radial mode decomposition. Correlations to varying mean flow speed and loading of the rotor blades are found. By means of an experimental variation, i.e. the installation of a honeycomb in the fan inlet, the dependence of the narrowband noise components on the lateral turbulence and the integral length scale is demonstrated. Comparisons with the analytical fan noise model give insight into the underlying dependencies. The model calculates the steady and unsteady flow field based on simplified state-of-the-art aerodynamic modelling. From this the strengths of the acoustic sources that are distributed radially along the individual blade spans are derived. The fast calculations allow extensive parameter studies, that help to trace back acoustic phenomena to their physical noise source mechanisms. The paper concludes with a comparison to results obtained for another test rig with similar dimensions. Both rigs show slightly different trends due to different blade designs.

lukas.klaehn@dlr.de

Practical Effect of Using Acoustically Untreated Test Ducts on the Non-Measurement Side, when Measuring Fan Sound Power Levels

Dario BRIVIO Regal Beloit Italy S.p.A. (Italy)

The measurement of fan sound power levels has been done in accordance with many national and international standards, like BS 848-2:1985, DIN 45635-38 and -9, AMCA 300:90 and ISO 5136. All these standards define the acoustical properties of the measurement environment, to identify the physical relationship between the sound pressure level, which can be measured, and the sound power level to be determined.

Unlike other noise sources, anyway, the inlet and outlet side of fans are acoustically connected, and the acoustical impedance of the environment on the non-measurement side influences the sound power level which can be observed on the measurement side. As a result, the repeatability of the sound power measurements on fans can be compromised using simple, acoustically untreated test ducts, on the non-measurement side.

A series of experimental measurements were carried out in a reverberating room, in accordance with the AMCA 300:90 standard, on the suction side of a centrifugal fan with scroll, connected to an airflow-measurement chamber with a simple “short outlet duct” on the discharge side of the fan.

Third-octave band sound power spectra where measured, at the same flow coefficients but at different fan speed values, with the test setup otherwise unmodified. The measured sound power levels were then reduced to “non dimensional sound power spectra”, as functions of “non dimensional frequencies”, according to the ISO 13348 computation method.

Satisfactory results could be achieved across most of the sound spectrum, but not around the blade-passing frequency and its second-order harmonic, i.e. where the fan is generating its major pure-tone components.

The blade-passing frequency amplitude is rising and dropping with the change of the fan speed and fundamental rotation frequency, and the variation of the amplitude can be related with the ratio between the BPF wavelength and the physical length of the simple, straight outlet duct, demonstrating to be a simple case of axial resonance along the outlet duct.

These test results confirm the importance of the new ISO 13347 family of standards, introducing the simplified anechoic termination of non-measurement ducts, to standardize the acoustic properties of the test setup, also on the non-measurement side.

Notwithstanding its merits, this method is not widely adopted by the industry, which is still struggling to find a practical solution to make measurements in all the many different duct sizes, required to cover an extensive fan range, with a still-acceptable investment and a manageable stockpile of different acoustically-treated test-ducts.

dario.brivio@regalrexnord.com

Investigation of a New Shielding Material for Slotted Tubes Regarding Sound Power Measurements of Ducted Fans

Jörg RIEDEL FAU Erlangen-Nürnberg (Germany)
Felix CZWIELONG FAU Erlangen-Nürnberg (Germany)
Stefan BECKER FAU Erlangen-Nürnberg (Germany)

Slotted tube probes are used in acoustic measurement technology, for instance, to measure the sound power level of fans and aggregates in duct flows. For this purpose, an approximately 450 mm long metal tube, which is slotted on one side, is screwed onto the actual measuring microphone. The slot can also be covered with a porous medium. The slotted tube acts as a turbulence screen and is supposed to suppress the flow-related hydrodynamic pressure fluctuations on the one hand and to let the acoustic pressure fluctuations, which are due to the fan, pass through to the microphone on the other. Aramid fabrics, sold under the trade name Kevlar®, are increasingly used as shielding material in aeroacoustic wind tunnels due to their material properties. In the present work, the question is investigated to what extent Kevlar® is suitable as a porous covering material for slotted tube probes. For this purpose, a standardized slotted tube probe is built, which is then covered with Kevlar®. The performance of the probe is then tested in an axial fan test rig. In addition, the performance of the probe is compared with an uncovered slotted tube probe. The probes are examined on the test rig at different operating points of the fan and turbulent inflow conditions that differ in terms of length scale and turbulence intensity. It is shown that the slotted tube probe covered with the aramide fabric can detect all acoustic characteristics of the axial fan, such as blade passing frequency and higher harmonics, in comparison to the uncovered probe, whereas the uncovered probe is not able to do so, especially at higher flow velocities. When comparing the probe covered with aramide fabric with a probe covered with a metal grid in an aeroacoustic wind tunnel, the turbulent inflow conditions are decisive for which of the two probes achieves a better signal to noise ratio. The results presented should serve to further improve slotted tube probes, which are often optimised in a university context.

ri@ipat.fau.de

A Round Robin Test of a Low-Pressure Axial Fan

Michail VOURAKIS Chalmers Institute of Technology (Sweden)
Mikael KARLSSON KTH Royal Institute of Tehcnology (Sweden)

We discuss the reproducibility of aerodynamic and aeroacoustic results on a benchmark low-pressure axial fan at different test facilities. The interest in this type of fans is growing due to the new demands for cooling of electrified powertrains in the automotive industry. But reference data sets for validating measurement methods and simulations are scarce. We will use the benchmark case set up and documented by Zenger et al [1]. They have published and made publicly available the geometry and test setups/methods to make it possible to reproduce their results. However, there are always practical difficulties; production differences when making the actual fan, variations in the lab setups and equipment for example.

We have 3D printed the shroud and three copies of the fan. They have then been painted/coated and manually sanded. The deviations of the final prototypes from the nominal geometry were measured in a standardized dimensional management set up. The surface roughness was also determined. The reason for printing three copies is to see the variations one can expect from small production differences.

Initially the fans were tested in a rig where accurate performance and aeroacoustic measurements can be made, but no detailed flow field measurements are available. In the benchmark case the upstream test chamber is anechoic, while in our rig the conditions are resembling that of a semi-anechoic room. We discuss the differences in the methods and results and how they can be interpreted and compensated for. The acoustic characteristic curve as well as spectra at different operating conditions are given along with discussions on directivity. Subsequent measurements in a rig which yields detailed flow field measurements were planned. However, non-conclusive results from the latter limit the discussion on results from the first testing facility.

[1] Zenger, F., Junger, C., Kaltenbacher, M., and Becker, S., "A Benchmark Case for Aerodynamics and Aeroacoustics of a Low Pressure Axial Fan", SAE Technical Paper 2016-01-1805, 2016, doi:10.4271/2016-01-1805.

michail.vourakis@chalmers.se

MEASUREMENTS OF AERO-ACOUSTIC INSTALLATION EFFECTS IN TANGENTIAL FAN

Joachim DOMINIQUE von Karman Institute (Belgium)
Fabrice AILLOUD Valeo Thermal System (France)
Olivier CHERIAUX Valeo Thermal System (France)
Manuel HENNER Valeo Thermal System (France)
Julien CHRISTOPHE von Karman Institute (Belgium)
Christophe SCHRAM von Karman Institute (Belgium)

Small ducted fans in HVAC applications are often affected by their integration within the final product. The flow and operating conditions induced by the fan working environment significantly impact its noise emission and aerodynamic performances. This paper presents a combined numerical/experimental approach using LES simulations and multi-ports measurements to investigate such installation effects for a centrifugal fan subjected to distorted inflow conditions. We observe that the distortion increases the fan broadband noise while suppressing its tonal emissions and we attribute those changes to the lack of constructive interference between individuals blades' forces in distorted inflows.

joachim.dominique@vki.ac.be

CONTROLLED INFLOW TURBULENCE - AEROACOUSTIC INTERACTION OF AXIAL FANS WITH AN ACTIVE TURBULENCE GRID

Felix CZWIELONG FAU Erlangen-Nürnberg (Germany)
Stefan BECKER FAU Erlangen-Nürnberg (Germany)

Leading edge noise is one of the dominant noise sources from axial fans. This sound source is particularly amplified under disturbed inflow conditions, i.e. high turbulence or inhomogeneous flow fields. In most technical applications, axial fans are operated in disturbed inflow conditions. The increased turbulence intensities and irregularities in the flow field are generated by other upstream system components. For example, heat exchangers and filters in the air-conditioning systems or protection grids and flow redirectors in the field of mobility are used on the suction side of the fan. The flow fields influenced by these components interact with the axial fan, which amplifies the leading edge noise. Since the inflow conditions differ from system to system, it is not possible to integrate this high amount of different flow profiles in the design process of the axial fan. For this reason, the axial fan is designed for undisturbed inflow conditions, which leads to increased sound emissions in real installation situations. In addition to this challenge, the lack of knowledge about the particular effects of turbulence quantities on the sound emissions of axial fans is a challenging task.

In most flows, turbulence characteristics such as turbulence intensity, length scale, anisotropy and inhomogeneity are highly connected. This made it difficult to investigate the influence of individual parameters and to gain a better understanding of the relation to the acoustics. The knowledge about the connection of sound generation mechanisms at the axial fan and turbulence characteristics would offer the possibility to estimate in advance the effects of different disturbed inflow conditions. In order to investigate these two tasks, an active grid was developed, which is used to generate defined inflow conditions. This offers the option to control the turbulence characteristics of the flow and thus to clarify the influence of individual turbulence characteristics. In addition, axial fans can be tested under different inflow turbulence conditions before they are installed in real systems. It was shown that with the active turbulence grid, the flow can be adjusted in a controlled manner and a large number of different flow conditions can be created. Based on the investigations, correlations between the turbulence characteristics and the sound emissions of axial fans were be concluded. In addition, it was shown that the mean flow fields of heat exchangers could be reproduced with this active grid.

cz@ipat.fau.de

INTRODUCING A NEW EFFICIENCY METRIC FOR LARGE-DIAMETER CEILING FANS – PRINCIPALLY HIGH-VOLUME, LOW-SPEED CEILING FANS

Christian TABER Big Ass Fans (United States)

The U.S. Department of Energy (USDOE) efficiency metric for large diameter ceiling fans (LDCF) was replaced by a new metric in 2021. LDCF are fans with a blade span (diameter) of 2.1 m (7.0-ft) and have been regulated In the United States since 2017 (test procedure) and 2019 (energy standard).

In a highly unusual development, a U.S. law was passed in 2020 to change the regulatory metric because the USDOE metric unintentionally capped the performance of LDCFs that had a high airflow utility relative to its blade span – i.e., larger-diameter fans can barely comply with the USDOE requirement even though they are highly efficient for their utility, and future evolution of these products was unfairly limited. The law led to a change in the U.S. Code of Federal Regulations in 2021 that put the CFEI metric into effect.

In legal documents, the USDOE metric was expressed only in inch-pound units of cubic-feet-per-minute (cfm) per watt (DOE cfm//W), averaged over time-weighted five test speeds, and with an integrated idle-fan standby-power measurement (the metric equivalent that is used in the Presentation Without Paper is cubic meters per second m^3/s). The cfm/W metric was replaced by Ceiling Fan Energy Index (CFEI), which is not time weighted, is calculated at two test speeds, and does not integrate an idle-fan standby-power measurement.

The CFEI metric has more stability over the range of fan operating speeds and provides a more accurate depiction of fan performance at higher airflows. CFEI also is less prone to abuse by bad actors that could meet efficiency requirements by simply slowing down an inefficient fan. Because the standby power measurement was eliminated, CFEI offers a more accurate representation of fan efficiency across its operating range.

CFEI is a unitless metric calculated from performance-rating tests conducted in accordance with the USDOE test procedure, which references ANSI/AMCA Standard 230-15, "*Laboratory Methods of Testing Air Circulating Fans for Rating and Certification*." There is no equivalent ISO standard to AMCA 230. CFEI is based on the Fan Energy Index (FEI) metric that was developed for commercial and industrial fans. FEI is a wire-to-air metric being used in the United States for energy codes used for buildings, and in state and federal fan regulations. FEI is being integrated into the ISO standards community as ISO 12759-6.

This presentation describes the CFEI metric, how it is calculated and applied. It also describes the ANSI/AMCA 230 test standard and updates the audience on revisions to the standard that are being considered during its current review/update cycle.

ctaber@bigassfans.com

Research-Based COVID Guidance for Large-Diameter Ceiling in Midsize Warehouses and Large Open Spaces

Michael IVANOVICH AMCA International (United States)
Liangzhu-Leon WANG Concordia University - Montreal (Canada)
Paul RAFTERY University of California, Berkeley (United States)
William BAHNFLETH Pennsylvania State University (United States)
Christian TABER Big Ass Fans (United States)

In August 2020, Air Movement and Control Association, International (AMCA) was made aware of COVID guidance on operation of circulating fans that was published by U.S. Centers for Disease Control of Atlanta (CDC), National Institute for Health (NIH), and other organizations. The guidance, which did not point to foundations stemming from research or prior practice, advised that circulating fans should be turned off.

AMCA met with several member companies that manufacture large diameter ceiling fans and the consensus position was that the guidance for turning fans off seemed wrong. Wouldn't mixing / destratification dilute concentrations in breathing zones? Staff contacted the NIH to discuss the guidance to learn about the provenance of the guidance and learned that the agency's subject matter experts would be consider changing the guidance if AMCA had better information. However, with no specific data in hand with which to respond, AMCA initiated a modeling-based research project for large-diameter ceiling fans operating in mid-size warehouses, which, especially in the context of the pandemic, was strategic given the need for more warehouses in the supply chains of online commerce.

Project Initiation: given the reasonable concern of conflict-of-interest or commercial bias from industry-funded research, AMCA assembled a bi-cameral project team consisting of a science team and an industry team. The industry team consisted of engineers from AMCA and member companies and the science team consisted of experts in computational fluid dynamics modeling, finite element analysis, infectious diseases, indoor air quality, and building science. The science team designed the study, reviewed the initial results and final report, and is participating in post-project peer-review journal publishing.

Modeling: the modeling approach, including assumptions, limitations, validation of the approach, and supplemental discussions are in the final report. The project used two 6-m (20-ft) AMCA-certified fans spaced 120-ft apart at three speeds: 0 m/s, 0.6 m/s (2 ft/s), and 6 m/s (10 ft/s). These speeds are average air speed exiting the fan. At 100 % speed provided an airflow of 90 m^3/s (192,000 cfm) at 78 rpm. Over 220 parametric simulations were run. The warehouse was modeled with and without racking. Zones were established relative to the workers under the fan so spaces in their vicinity (working zone) and outside their vicinity (breathing zone) could be investigated. Worker and their orientation relative to the fans were varied.

Results: the options for operating the fan (different speed or direction), operating the fans at the highest feasible speed consistently outperforms the other options. However, operating fans at the maximum speed generates high airspeeds in the occupied zone, which will not be practical in some conditions, for example, as it would cause thermal discomfort in cold indoor conditions. Where lower airspeeds are preferred: Fan speed at 3 m/s (10 ft/s) in upward-flow direction is a good option, although it is not as good as 3 m/s (10 ff/s) with downward flow. Reversing fans at higher fan speed (3 m/s) reduces the performance compared to high speed with downward flow. Reversing fans at lower fan speed (e.g., 0.6 m/s (~ 2.0 ft/s)) may reduce the whole warehouse airflow speed and thus lower diluting effect, so the whole warehouse concentration could become higher. Thermal plumes from workers are essential, especially for still air cases; local air mixing and movement could help to dilute and reduce the chance of creating local hotspots.

AMCA has developed COVID guidance for large diameter ceiling fans for warehouses based on these results. The final report and input files for modeling are available at https://bit.ly/COVID_LDCF.

mivanovich@amca.org

EVALUATION AND APPLICATION OF AIR CURTAIN EFFECTIVENESS METHODOLOGY

David JOHNSON Berner International LLC (United States)

Air curtains have been widely used on building entrances as a barrier to reduce energy losses due to infiltration/exfiltration while still permitting unobstructed passage of pedestrian traffic. Many building energy codes are published that prescribe the requisite for energy efficient designs. These codes address building entrances that separate conditioned space from the exterior with a requirement that they shall be protected with an enclosed vestibule.

Recent studies examine and validate the performance or effectiveness of air curtains at a macro or whole building perspective and compare that to vestibules. The studies illustrate that several factors impact air curtain effectiveness including air discharge velocity, angle, door height and width, indoor and outdoor pressure conditions. Existing test methods (and associated standards/codes) evaluate the effectiveness based on:

1.) either the aerodynamics characterization of air curtain through the measurements of air curtain jet velocity distribution and degradation,

2.) or the measurements of infiltration/exfiltration rates with/without the air curtain under different operating conditions, i.e. pressure difference across the door.

The 1st method is easier to be conducted but is not directly related to evaluating the capability of reducing infiltration/exfiltration whereas the 2nd method can do so by testing each specific unit, which is often costly, time-consuming and sometimes impractical. Therefore, a relatively simpler method is required to combine both methods for the evaluation of air curtain effectiveness during its design, selection and operation.

This study aims to develop such a calculation method to relate the aerodynamics performance test method in ANSI/AMCA Standard 220-05 or ISO 27327-1:2009 directly to evaluating the air curtain's capability of reducing infiltration/exfiltration rates. It is proposed that an air curtain discharge be separated into several sub-sections to consider the diversity of the discharge velocity along the air curtain width. Case studies are provided for the demonstration of the whole process. A validation study using the previous experimental results is also conducted. In comparison to the existing effectiveness test methods, the new method is a more feasible solution to various air curtain products and installation scenarios.

djohnson@berner.com

Exhibitors

turbo designsuite
by Advanced Design Technology

amca INTERNATIONAL

CETIAT

CARNOT

cetim

CFturbo

DASSAULT SYSTEMES

ep-e.com

HEAD acoustics

HEXAGON

m+p international

Polytec

systemair

fan2022

EP Ehrler Prüftechnik Engineering GmbH

We are the leading solution provider in Germany in the field of flow (air and other gases), pressure and temperature control, leak testing and the high-precision measurement and control technology of gases (e.g. natural gas).

The degree of energy efficiency of various heating, ventilation or air conditioning systems is the decisive yardstick for operating costs and environmental friendliness. EP-E supports you in the functional testing of your components such as heat pumps, heat exchangers, gas heating systems, ventilation devices, refrigeration machines as well as valves, controllers and sensors in compliance with the required standards. We build test benches that are tailored to your requirements and products.

EP-E calibration standards are transfer standards with the highest possible measurement uncertainty, the measurement results of which can be traced back directly to the SI units m and s. Over the past few years, we at EP-E have designed a range of transfer standards that are designed for use in calibration laboratories.

Our fan / blower test benches are used to measure and record the characteristic curves of fans and ventilators according to DIN EN ISO 5801 and AMCA 210. The individual test benches can be designed for a wide variety of volume flow ranges from 1.5 to 100,000 m³/h, either as suction or pressure side chamber test benches. The chamber pressure can be designed individually. This makes it possible to simulate different load conditions on the test item. The use of specific measuring elements enables a measuring accuracy of up to 0.5% MW.

Perfection in fluids.
The right *flow* by German engineering.
ep-e.com

About HEAD acoustics

HEAD acoustics GmbH is one of the world's leading companies offering holistic sound and vibration analysis solutions. In the telecom sector, the company enjoys global recognition due to its expertise and pioneering role in developing hardware and software for measuring, analyzing, and optimizing voice and audio quality, as well as customer-specific solutions and services. HEAD acoustics' range of services covers sound engineering for technical products, investigation of environmental noise, speech quality engineering, consulting, training, and support. The medium-sized company from Herzogenrath near Aachen has subsidiaries in China, France, Italy, Japan, South Korea, the UK, the USA, and numerous sales partners worldwide. At FAN 2022, you will experience these products and technologies:

- ArtemiS SUITE is a modular software platform for measuring and analyzing sound and vibration with the aim of optimizing product and environmental noise. We will show its application on a notebook using a ventilation noise example metric.
- With SQala, jury tests can be set up quickly and easily and performed either at a single-user system or in a listening studio. SQala supports playback of binaural recordings.
- HSU III.3: analog low-noise artificial head with special condenser microphones that enable aurally accurate recordings of extremely low sound pressure levels such as those occurring in IT. The low-noise artificial head boasts a dynamic range of 106.5 dB(A) with extremely low inherent noise.
- *lab*HSU: *lab*HSU is an extraordinarily flexible dual-channel data acquisition system in the format of the HEAD*lab* family. It can be operated as a frontend on a computer, as a module in a HEAD*lab* system or as a stand-alone system by remote control via smartphone, tablet, or the RC X remote control.
- Binaural playback with *lab*P2 and headphones: *lab*P2 is a binaural headphone equalizer (equalization: FF, ID, DF, USER) that enables aurally accurate playback of binaural recordings via one or two separately equalized headphones. It can be used in USB mode with a PC or as part of a HEAD*lab* system.
- SQuadriga III & SQobold: Mobile, modular, and scalable multi-channel recording, online analyses (e.g., loudness, sharpness) and playback systems.

m+p international

Experts in Vibration

Acoustic, Vibration
Data Acquisition
Test Stand Engineering
Measurement Harware

Customers / market segments

m+p international is a leading global supplier of vibration controllers for shaker testing. We also provide analyzers for noise and vibration measurement, analysis and reporting. Suit all your NVH needs in the field and in the lab: structural dynamics, modal analysis, rotating machinery analysis, acoustics, FFT analysis, time history recording.
Some references in France: Aerospace (Airbus, Thales, Safran, Dassault...), Defense (Safran, Thales, DGA TT, Etienne Lacroix...), Automotive (PSA, Faurecia, Valeo, Continental, Autoliv, ACS...), Universities and Research center (CNAM, ENSAM, Centrale Lyon, INSA Blois, IUT de Saint-Etienne, ESIX, Femto-ST...).
Many other companies worlwide already trust m+p products for their NVH testing.

DOMAINES D'EXPERTISE :
Acoustique, Vibration
Acquisition de données
Bancs d'essais, Moyens d'essais

Clients / Marchés

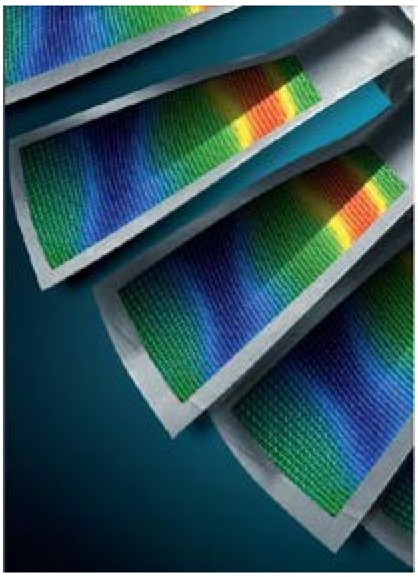
M+p international est l'un des principaux fournisseurs mondiaux de contrôleurs de vibration pour pots vibrants. Nous fournissons également des analyseurs pour la mesure, l'analyse et la création de rapports sur le bruit et les vibrations.
Voici les domaines que nous pouvons vous proposer afin de répondre à vos besoins NVH sur le terrain et en laboratoire : dynamique des structures, analyse modale, analyse des machines tournantes, acoustique, analyse FFT, enregistrement des signaux temporels.

Quelques références en France : Aéronautique (Airbus, Thales, Safran, Dassault...), Défense (Safran, Thales, DGA TT, Etienne Lacroix...), Automobile (PSA, Faurecia, Valeo, Continental, Autoliv, ACS.. .), Universités et Centre de Recherche (CNAM, ENSAM, Centrale Lyon, INSA Blois, IUT de Saint-Etienne...).
De nombreuses autres entreprises dans le monde font déjà confiance aux produits m+p pour leurs tests NVH.

Présentation Polytec France

Le groupe Polytec est présent dans le monde entier grâce à ses filiales et représentants. Fabricant et spécialiste, Polytec conçoit des équipements de mesure optique pour le contrôle en temps réel non destructif : vibrométrie, vélocimétrie, métrologie de surface, spectrométrie en ligne.
Également distributeur, Polytec propose des gammes de produits complémentaires en photonique, vision, procédés industriels, acoustique … Polytec accompagne l'ensemble des professionnels du domaine Public (Education, Recherche) et de l'Industrie quelque soit le domaine : Aéronautique, Automobile, Défense, Electronique, Agroalimentaire, Biomédical, Agriculture, Cosmétique… Implantée en région Parisienne, la filiale française intervient sur l'ensemble du territoire et propose de multiples alternatives en plus d'un service technique de qualité : vente, location, prestation de mesure, formation, calibration...

Spécialiste des vibromètres laser, utilisés pour des applications de caractérisation vibratoire et vibro-acoustique, ces instruments sont des atouts clés dans tout le cycle de développement de vos produits, de la conception (analyse modale, corrélation calcul/essais) au contrôle qualité en fin de ligne de production.



Sponsors

Platinum sponsor & Gala Dinner: ebm papst

Bachmühle 2
74673 Mulfingen, Germany
Website: www.ebmpapst.com

Gold sponsor: AMCA

30 West University Drive
Arlington Heights
IL 60004 , USA
E-mail: mstevens@amca.org
Website: www.amca.org

Journal Issue: Ziehl Abegg

Heinz-Ziehl-Straße
74653 Künzelsau, Germany
Website: www.ziehl-abegg.com

Delegate bags: ADT (Turbo Design)

Dilke House
1 Malet Street, London, UK
WC1E 7JN
Phone: +44 (0)20 7299 1170
E-mail: sales@adtechnology.co.uk
Website: www.adtechnology.co.uk
AGS Consulting

Lanyards: Systemair

Seehöfer Straße 45
97944 Boxberg
E-mail: info@systemair.de
Website: www.systemair.com

Co-organizers

CETIAT

Centre Technique des Industries Aérauliques et Thermiques Domaine Scientifique de la Doua,
25 avenue des Arts, BP 52042
69603 Villeurbanne Cedex
France
François Battistoni, Sales engineer
Phone: +33 472 44 49 00
E-mail: commercial@cetiat.fr
Website: www.cetiat.fr

CETIM

Technical Centre for Mechanical Industries
52 Avenue Félix Louat, BP 80067
60304 Senlis
France
Céline Cammarata
Phone: +33 (0)9 70 82 16 80
E-mail: contact@fan2022.org
Website: www.cetim.fr
www.cetim-engineering.com

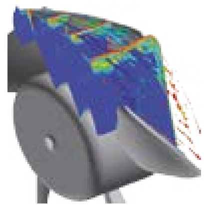
TURBOMACHINERY DESIGN AND OPTIMIZATION with

turbo designsuite
by Advanced Design Technology

Design of Axial Fans with Increased Efficiency and Reduced Noise

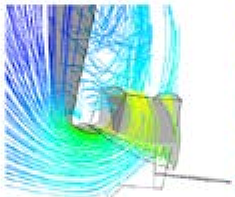
In the design of axial fans, the tip leakage effects have a major impact on efficiency and noise. By using TURBOdesign1, and correct choice of spanwise swirl distribution and streamwise loading, the leakage flow was controlled leading in reduction in noise and increase in efficiency.

Baseline Design

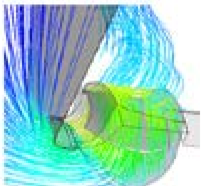


Inverse Design

Design of Centrifugal Fan to Improve Efficiency at Multiple Operating Points



Baseline Design



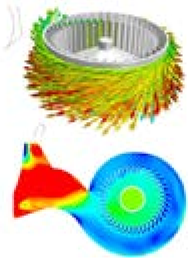
Inverse Design

In design of centrifugal fans, improving efficiency at multiple operating points is a key requirement. TURBOdesign Suite enables rapid optimization of centrifugal fans for multiple operating points by its efficient surrogate model based optimization system.

Design of Centrifugal Blower Stages

TURBOdesign1 and TURBOdesign Volute can be used to design rotor and casing of centrifugal blowers. In this example, the redesign of the complete blower by TURBOdesign Suite resulted in 10% increase in stage total-static efficiency.

Baseline Design



Inverse Design

UNIQUE ADVANTAGES OF 3D INVERSE DESIGN

Design Process
Powerful design and optimization environment integrated seamlessly into your company workflow.

Efficiency
Typical improvements of 3% to 5% efficiency over state-of-the-art designs.

Faster
Reduced design time and computational costs by 30% across all projects.

Optimization
Enabling automatic optimization by reducing computational costs by up to 10x.

+44 (0)20 7299 1170
sales@adtechnology.com
www.adtechnology.com

30 Millbank
London SW1P 4DU
United Kingdom

Advanced Design Technology

fan2022

Notes

Sponsors

Co-organizers

2112-013