fan2018

BOOK OF ABSTRACTS

INTERNATIONAL CONFERENCE ON FAN NOISE, AERODYNAMICS, APPLICATIONS AND SYSTEMS

18 to 20 April 2018 in Darmstadt, Germany

www.fan2018.org

Organized by

cetim

CETIAT ensemble, innover et valider

UNIVERSITÄT SIEGEN

TECHNISCHE UNIVERSITÄT DARMSTADT

CONTENTS

INTRODUCTION FROM THE CONFERENCE CHAIR

Welcome to Fan 2018.

The Fan 2018 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. Panel sessions will also give you the chance to learn more about energy efficiency regulation, market surveillance and fan performance verification tools.

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 2018.

Darmstadt is a city in the state of Hesse in Germany, located in the southern part of the Rhine-Main-Area. Darmstadt holds the official title "City of Science" as it is a major center of scientific institutions, universities, and high-technology companies. Darmstadt was formerly the capital of a sovereign country, the Grand Duchy of Hesse and its successor, the People's State of Hesse, a federal state of Germany. As the capital of an increasingly prosperous duchy, the city gained international prominence and remains one of the wealthiest cities in Europe. In the 20th century, industry, as well as large science and electronics sectors became increasingly important, and are still a major part of the city's economy.

This is a particularly important time for the air movement community. Fan efficiency is regulated in Europe, but US Department of Energy (DoE) regulatory efforts have stalled indefinitely. Meanwhile, California is unilaterally developing its own fan efficiency “appliance” regulation” that is not within a building energy code. Other states, including New York, have communicated their intent to regulate equipment, so it’s quite possible that California will set a precedent for other states. In a worst-case situation, other states will regulate fans differently from California. 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 2018 you take an important step towards embracing the regulatory challenges our community faces. I look forward to welcoming you to Fan 2018.

I would also like to take this opportunity for and on behalf of Fan 2018 organizing committee to welcome you to Darmstadt, and to thank you for attending Fan 2018. 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 2018 Organizing Committee

SPONSORS

EXHIBITORS

ORGANIZING CONSORTIUM

CETIAT, France (Alain GUEDEL, François BESSAC)

Cetim, France (Xavier CARNIEL)

Universität Siegen, Germany (Thomas CAROLUS)

Technische Universität Darmstadt, Germany (Peter PELZ)

For the detailed information about the members of the organizing consortium see page 10 ff.

ORGANIZING COMMITTEE

Geoff SHEARD (AGS Consulting LLC, USA) - Conference Chairman

Jürgen SCHÖNE (ebm-papst, Germany) - Conference Vice-Chairman

Alain GUEDEL (CETIAT, France) - Fan Noise track co-chair

Xavier CARNIEL (Cetim, France) - Fan noise track co-chair

Alessandro CORSINI (Sapienza University of Rome, Italy) - Fan Aerodynamics track co-chair

Thomas CAROLUS (Universität Siegen, Germany) - Fan Aerodynamics track co-chair

Peter PELZ (Technische Universität Darmstadt, Germany) - Fan Application and Systems track co-chair

Mark STEVENS (AMCA, USA) - Fan Application and Systems track co-chair

François BESSAC (CETIAT, France)

Mark BUBLITZ (The New York Blower Company, USA)

Thomas DAMM (VDMA, Germany)

Christine KRAUSE (Universität Siegen, Germany)

Daniela KALLER (Technische Universität Darmstadt, Germany)

Sylvia TRUCKENMUELLER (ebm-papst, Germany)

SCIENTIFIC ADVISORY COMMITTEE

SUPPORTING ORGANISATIONS

AMCA International:
Air Movement and Control Association International

ebmpapst:
Fan manufacturer

EUROVENT:
European Committee of Air Handling and Refrigeration Equipment Manufacturers

FETA:
Federation of Environmental Trade Associations

FMA:
Fan Manufacturers Association

HEVAC:
Heating, Ventilation and Air-Conditioning Manufacturers Association

Uniclima:
French association representing the air-handling, cooling, heating and refrigeration industry (Syndicat des industrie Thermiques, Aérauliques et Frigorifiques)

VDMA:
Verband Deutscher Maschinen und Anlagenbau (German Engineering Federation)

ABOUT THE ORGANIZING CONSORTIUM

CETIAT is a French organization which provides research, product development, testing and calibration service in the air-handling, heating and acoustic fields. 140 employees work in this laboratory including 50 test rigs located in Lyon, France.

CETIAT has an extensive and practical knowledge of the aerodynamics and acoustics of fans, mainly axial and centrifugal fans. In this field, as in others, CETIAT carries out research programs (technical know-how compilation, design tool development, capitalisation of expertise) and is strongly involved in fan standards development. This skill and expertise are used to offer services to the companies and manufacturers concerned.

Its customers are involved in the Heating, Ventilating and Air-Conditioning industry, but also come from other fields of activity where CETIAT's skill may be useful such as the automotive and railway industries for instance.

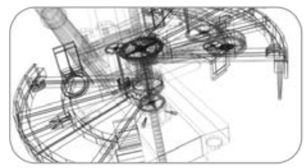
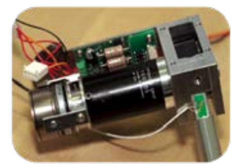
CETIAT provides assistance to its customers at all stages of their product or process development:

- Staff training,
- Technological survey,
- Design and integration of fans using CFD and house databases,
- Measurement of air and sound fan performance in accordance with ISO 5801, ISO 13347, ISO 5136, AMCA 210, AMCA 300, …
- Calibration of measurement sensors.

www.cetiat.fr

Cetim
The French technological institute for mechanics

New concepts in mechanical engineering

As the leading French player in the fields of mechanical engineering innovation and R&D, Cetim has built up a wide network of partners. Its engineers and technicians operate in more than 30 countries each year.

R&D function is carried out either within specific sectors or cutting across sector boundaries, and within either a national or an international context. It embraces a range of complementary aspects, including prospective studies in conjunction with international scientific communities, R&D concerning all areas of mechanical engineering, industry-specific studies and projects, and the large-scale federative technological projects.

Cetim provides a comprehensive array of services to the mechanical engineering industry from consulting to testing and from engineering to training in new skills.

Cetim is a member of the Carnot Institutes network and a founding member of the French Alliance for the Industry of the Future.

Noise and Vibration Engineering : one of the Cetim's areas of expertise

For this area, our mission is to conduct experimental and numerical studies in the following fields:

- structural dynamics, diagnosis, monitoring and vibration expertise (rotating machines, industrial facilities…)
- noise and vibration reduction in industrial environment, vibro-acoustic design and expertise, machines and equipment certification

These studies are performed as well for mutualized actions (works requested and steered by representatives of each profession in the mechanical industry, support to SME'S and R&D) as for commercial services.

CONTACT
Céline Cammarata
Tél. : 33(0)344673682
sqr@cetim.fr

1803-034

The Chair of Fluid Systems at the Technische Universität Darmstadt, headed by Prof. Dr.-Ing. Peter Pelz, has 29 staff members including scientific staff and the mechanical workshop. The funding was 1.75 million Euros in 2017. The Chair was founded in 1897 and has a profound experience in theoretical and experimental method for design, assessment and validation of all kinds of turbomachinery operating with air, water or oil. This includes seals, bearings, sensors and drives.

Product validation by model test, scaling methods and model testing at full scale Reynolds number using a pressure chamber is one of the research fields. Further fields of research are system optimization by algorithmic system design using TOR (Technical Operations Research) and techno-economic assessments of turbomachinery in different environments. In recent years, the digitalization is getting more and more important as shown by projects regarding predictive scenarios using soft sensors as well as research and lab data management.

Predictive Scenarios

soft sensors for uncertain gas
+ physical based

soft sensors for surge detection
+ kinematic stall
+ dynamic stall

Techno-Economic Assessment

system characterization
+ considering technical and economical issues

Data Management

storing measurement data
+ standardized
+ fast application

Product Qualification

pressure chamber
+ model test at full scale Reynolds number

scaling methods
+ axial and centrifugal fans
+ positive displacement pumps
+ turbocharger

Algorithmic System Design

system design
+ usable for all kinds of ventilation systems

system optimization
+ optimal topology
+ suitable components
+ combined with scaling methods

Institut für Fluidsystemtechnik
Prof. Dr.-Ing. Peter F. Pelz
Otto-Berndt-Str. 2
64287 Darmstadt
Tel.+49 6151 16 - 27100
Fax+49 6151 16 - 27111
sekretariat@fst.tu-darmstadt.de
www.fst.tu-darmstadt.de
www.tu-darmstadt.de

Institute
Fluid- and Thermodynamics
Prof. Dr.-Ing. Th. Carolus

Steinbeis-Transferzentrum
Strömungstechnik und
Strömungsmaschinen

The Applied Fluid Mechanics and Turbo Machinery Group in the Institute of Fluid- and Thermodynamics of the University of Siegen Germany is headed by Prof. Dr.-Ing. Thomas Carolus. The group has more than 30 years of expertise in design, analysis and optimization of turbo machinery, e.g. fans and turbines for air and water. A further focus is on the analysis and design of systems with these turbo machines, for instance flow induced vibrations in air delivering systems, energy yield of maritime ocean energy systems, etc.

Efficiency and low noise emissions are the two main targets in nowadays turbo machinery and in particular fan design. Fast analytical design methods, Computational Fluid Mechanics (CFD) simulations utilizing Reynolds-averaged Navier-Stokes, large eddy and Lattice-Boltzmann methods are essential tools that have been developed, refined and validated during the last years. With the advent of new manufacturing methods (3D printing, fibre reinforced materials) optimization plays an increasing role. Based on advanced design of experiment (DoE) methods, numerical simulations and meta-models utilizing artificial neural network strategies, fast and efficient optimization work flows have been developed. Recently, sound quality evaluation of fan and fan-related sound via psychoacoustic methods has become a new field of research. Own aerodynamic and acoustic test rigs according to relevant standards and a professional hearing laboratory are available, manufacture of precise prototypes is done in an own workshop. The high performance computer cluster of the university is available for simulations with a high demand of computational resources.

The group widely publishes the results of its research and continuously presents the results at national and international meetings and invited lectures. So far 180 publications in archival journals and conference proceedings are available.
(http://www.mb.uni-siegen.de/iftsm/forschung/veroeffentlichungen.html).

Dr. Carolus is Professor of Applied Fluid Mechanics at the University of Siegen, a position he has held since 1990. He holds Masters degrees in Mechanical Engineering from the Technical Hochschule Karlsruhe (now Karlsruhe Institute of Technology, KIT) and the Georgia Institute of Technology, as well as a Ph.D. in Engineering from the Technical Hochschule Karlsruhe. Following his studies he spent four years as Group Manager at Bosch GmbH. He teaches and conducts research in the general areas of aerodynamics, acoustics and aeroacoustics focusing on turbo machinery, fluid power and fluid systems.

Know-how and technology transfer into industry is mainly managed via a Steinbeis transfer enterprise. Steinbeis is a German service provider in the field of knowledge and technology transfer with more than 1,400 transfer enterprises in all fields. In cooperation with the University of Siegen the Steinbeis transfer enterprise "*Strömungstechnik und Strömungsmaschinen*", headed by Prof. Carolus, is able to deliver client projects not just confidentially and reliably, but also on time.

www.uni-siegen.de

LIST OF SESSIONS

Keynote Lectures

Turbomachinery - Past and Future of CFD Simulations

- SCHEUERER Georg – ISimQ GmbH (Germany)

Standards and Regulation

- BREEN Tony – Nuaire (United Kingdom)

Gotthard Tunnel Ventilation System Design

- BRANDER Christoph – Pöyry Schweiz AG (Switzerland)

Wednesday 18 April 2018

A1 Fan Design & Materials

FLT Working Group Fans - Benefits and Results of Pre-Competitive Community Research

- REICHERT Erik – ebm-papst Mulfingen GmbH & Co. KG (Germany)

Endless Fibre Reinforced Composite-Metal-Impeller: Material Related Design and Dimensioning Process for Hybrid Radial-Fans

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

Integrative Simulation Method for the Prediction of Anisotropic and Time-Dependent Mechanical Behavior of Injection Molded Fiber-Reinforced Fan Impellers - Numerical and Experimental Approach

- LASS Andre – University of Rostock (Germany)
- HERTLE Sebastian – Universty of Erlangen (Germany)
- WURM Frank-Hendrik – University of Rostock (Germany)
- DRUMMER Dietmar – Universty of Erlangen (Germany)

Integrative Simulation Method for the Prediction of Anisotropic and Time-Dependent Mechanical Behavior of Injection Molded Fiber-reinforced Fan Impellers - Creep Modelling

- HERTLE Sebastian – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- LASS Andre – Universität Rostock (Germany)
- DRUMMER Dietmar – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- WURM Frank-Hendrik – Universität Rostock (Germany)

A2 Noise Generation Mechanisms

Potential Noise Source Identification in a Single Stage Rotor-Stator Compressor

- HARLEY Peter – Dyson (United Kingdom)
- CORRALEJO Raul – Dyson (United Kingdom)
- COLLISON Michael – Dyson (United Kingdom)
- LIRVAT Jimmy – Dyson (United Kingdom)
- WILSBY Oscar – University of Cambridge (United Kingdom)

Sound Radiation of a Smoke-Removal Fan System

- PASCO Yann – Université de Sherbrooke (Canada)
- MOREAU Stéphane – Université de Sherbrooke (Canada)

Investigation on the Noise of an Axial Low Mach-Number Stage with a Heterogeneous Stator

- PESTANA Miguel – Safran Aircraft Engines, École Centrale de Lyon, Université de Sherbrooke (France)
- SANJOSÉ Marlène – Université de Sherbrooke (Canada)
- MOREAU Stéphane – Université de Sherbrooke (Canada)
- ROGER Michel – École Centrale de Lyon (France)
- GRUBER Mathieu – Safran Aircraft Engines (France)

Tip Leakage Flow and its Implication on the Acoustic Signature of a Low-Speed Fan

- MARSAN Aurélien – Université de Sherbrooke (Canada)
- DOMINIC Lallier-Daniels – Université de Sherbrooke (Canada)
- SANJOSÉ Marlène – Université de Sherbrooke (Canada)
- MOREAU Stéphane – Université de Sherbrooke (Canada)

A3 Application of Analytical, Computational and Experimental Methods I

Design and Investigation of a Multistage Axial Contra-Rotating Fan

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

Parametric Study of Volutes for Optimal Centrifugal Fan Impellers

- GJETA Ardit – Polytechnic University of Tirana (Albania)
- BAMBERGER Konrad – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)
- LONDO Andonaq – Polytechnic University of Tirana (Albania)

Blade Design and Optimization of Low Pressure Axial Fans using a Cascade Approach

- WINTERBERG Heiko – Volkswagen AG (Germany)
- FRIEDRICHS Jens – Technische Universität Braunschweig (Germany)

Flow Loss and Flow Structure of Circular Arc Blades with Different Leading and Trailing Edges

- BIAN Tao – Jianghan University (China)
- LANDFESTER Christian – Technical University Kaiserslautern (Germany)
- FENG Jun – Jianghan University (China)
- HAN Qianpeng – Jianghan University (China)
- BÖHLE Martin – Technical University Kaiserslautern (Germany)

B1 System & Preliminary Design

Analytical Approach for the Performance Prediction of High Pressure Centrifugal Fans in Serial Arrangements

- NUDISCHER Martin – University of Stuttgart (Germany)
- BINZ Hansgeorg – University of Stuttgart (Germany)
- BACHMANN Matthias – University of Stuttgart (Germany)
- RECKER Stefan – University of Stuttgart (Germany)

Malfunctioning of Air-Delivering Systems - Examples of Fan Driven Transient and Oscillating Flows

- CAROLUS Thomas – University of Siegen (Germany)
- HOMRIGHAUSEN Bernd – University of Siegen (Germany)

Technical Operations Research (TOR) - Algorithms, not Engineers, Design Optimal Energy Efficient and Resilient Cooling Systems

- LEISE Philipp – Technische Universität Darmstadt (Germany)
- ALTHERR Lena – Technische Universität Darmstadt (Germany)
- PELZ Peter – Technische Universität Darmstadt (Germany)

Softsensor for the Characterisation of the Process Fluid

- SCHMITZ Christian – Technische Universität Darmstadt - Industrial Science GmbH (Germany)
- NAKHJIRI Mehdi – Industrial Science GmbH (Germany)
- PELZ Peter – Technische Universität Darmstadt (Germany)

Extended Rankine-Betz Theory for Design of Tunnel Ventilation Systems

- PELZ Peter – Technische Universität Darmstadt (Germany)
- SAUL Sebastian – Technische Universität Darmstadt (Germany)
- DOBERSTEIN Christopher – Technische Universität Darmstadt (Germany)

B2 Noise Prediction by Analytical or Numerical Models I

Investigation of Different Fan Noise Prediction Methods

- JUNGER Clemens – Technische Universität Wien (Austria)
- KALTENBACHER Manfred – Technische Universität Wien (Austria)

Numerical Analysis of the Acoustic Field of an Axial Fan at Disturbed and Undisturbed Inflow Conditions

- ALAVI MOGHADAM Seyed Mohsen – RWTH Aachen University (Germany)
- POGORELOV Alexej – RWTH Aachen University (Germany)
- ROIDL Benedikt – RWTH Aachen University (Germany)
- MEINKE Matthias – RWTH Aachen University (Germany)
- SCHRÖDER Wolfgang – RWTH Aachen University (Germany)

Numerical Aeroacoustic Analysis of Small Cooling Fans for Electronic Devices

- TALBOT Alexis – Free Field Technologies (Belgium)
- DETANDT Yves – Free Field Technologies (Belgium)
- HATANAKA Shogo – Software Cradle Co., Ltd (Japan)
- HIRAI Kenichiro – Software Cradle Co., Ltd (Japan)
- MINARIKAWA Gaku – Hosei University (Japan)

Observation of Noise of Fan with Obstacle for Electric Components

- KANEKO Kimihisa – Fuji Electric Co., Ltd. (Japan)
- MATSUMOTO Satoshi – Fuji Electric Co., Ltd. (Japan)
- YAMAMOTO Tsutomu – Fuji Electric Co., Ltd. (Japan)

Unsteady Simulation of Tonal Noise from Isolated Centrifugal Fan

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

B3 Case Studies: Aerodynamics I

Development of High Efficiency Fan System for Outdoor Unit of Air-conditioner

- IWASE Taku – Hitachi, Ltd., R&D Group (Japan)
- KATAYAMA Erika – Hitachi, Ltd., R&D Group (Japan)
- SEKIYA Sachio – Hitachi, Ltd., R&D Group (Japan)
- KISHITANI Tetsushi – Hitachi-Johnson Controls Air Conditioning, Inc. (Japan)

Influence of the Enclosure on the Performance of Radial Fans

- KLEPP Georg – OWL University of Applied Science (Germany)
- KAMPHAUSEN Walter – OWL University of Applied Science (Germany)

Rapid Prototyping of the Small Scale CSP Fan

- VOLPONI David – Sapienza University of Rome (Italy)
- WILKINSON Michael, B. – Stellenbosch University (South Africa)
- DELIBRA Giovanni – Sapienza university of Rome (Italy)
- CORSINI Alessandro – Sapienza University of Rome (Italy)
- VAN DER SPUY Sybrand, J. – Stellenbosch University (South Africa)
- VON BACKSTRÖM Theodor W. – Stellenbosch University (South Africa)

A Study on Design Optimization of Centrifugal Fan of High Voltage Generator by Numerical Analysis

- JANG Chungman – Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)
- LEE Joonyeob – Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)
- JEON Moojong – Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)

Thursday 19 April 2018

C2 Experimental Methods for Localizing / Characterizing Sources I

Study of Sound Source Distribution in Centrifugal Fan Using Transparent Wall

- MIN SWE War War – Mandalay Technological University (Myanmar (Burma))
- OKUMURA Tetsuya – Nagasaki University (Japan)
- KOYAMA Atsuhiro – Nagasaki University (Japan)
- HAYASHI Hidechito – Nagasaki University (Japan)
- HIRATA Makoto – Nagasaki University (Japan)
- NOGAMI Wakawa – Panasonic Eco Systems (Japan)

Comparison of Multiple Fan System Assemblies Using an Acoustically Transparent Duct

- TOKAJI Kristóf – Budapest University of Technology and Economics (Hungary)
- SZEKER Balázs – Budapest University of Technology and Economics (Hungary)
- HORVÁTH Csaba – Budapest University of Technology (Hungary)

Comparison of Microphone Array Methods for the Characterization of Rotating Broadband Noise Sources

- OCKER Christof – Aalen University (Germany)
- HEROLD Gert – Technische Universität Berlin (Germany)
- KRÖMER Florian – Friedrich-Alexander University Erlangen-Nürnberg (Germany)
- PANNERT Wolfram – Aalen University (Germany)
- SARRADJ Ennes – Technische Universität Berlin (Germany)
- BECKER Stefan – Friedrich-Alexander University Erlangen-Nürnberg (Germany)

C3 Application of Analytical, Computational and Experimental Methods II

Aeroacoustic and Aerodynamic Performance of Silencer Units at Different Inflow Profiles

- WALTER Johannes – Karlsruhe Institute of Technology (Germany)
- GABI Martin – Karlsruhe Institute of Technology (Germany)

CFD Analysis of Axial Flow Fans for CSP Air-Cooled Condensers

- VOLPONI David – Sapienza University of Rome (Italy)
- WILKINSON Michael – Stellenbosch University (South Africa)
- VAN DER SPUY Johan – Stellenbosch University (South Africa)
- BONANNI Tomasso – Sapienza Univerisity of Rome (Italy)
- TIEGHI Lorenzo – Sapienza University of Rome (Italy)
- DELIBRA Giovanni – Sapienza University of Rome (Italy)
- CORSINI Alessandro – Sapienza University of Rome (Italy)
- VON BACKSTRÖM Theodor W. – Stellenbosch University (South Africa)

OpenFOAM Application on Performance Predictions of an Industrial Centrifugal Fan with Airfoil Blades

- WANG Jing – Daltec Process Fans (Canada)
- KAMUTZKI Marcel – Daltec Process Fans (Canada)

D2 Experimental Methods for Localizing / Characterizing Sources II

Separation of Tonal and Broadband Noise Components by Cyclostationary Analysis of the Modal Sound Field
in a Low-speed Fan Test Rig

- BEHN Maximilian – German Aerospace Center (DLR) (Germany)
- PARDOWITZ Benjamin – German Aerospace Center (DLR) (Germany)
- TAPKEN Ulf – German Aerospace Center (DLR) (Germany)

Energy Dissipation in the Axial Fan Tip Clearance Flow

- MILAVEC Matej – Hidria Rotomatika d.o.o. (Slovenia)
- PIVK Stanislav – Hidria Rotomatika d.o.o. (Slovenia)
- VIDAL DE VENTOS Daniel – Hidria Rotomatika d.o.o. (Slovenia)
- ŠIROK Branko – Faculty of Mechanical Engineering Ljubljana (Slovenia)
- BIZJAN Benjamin – Abelium d.o.o. (Slovenia)

A Cyclostationary Approach in Extracting Modulation Feature from Fan Vibrations

- CHU Ning – Zhejiang University (China)
- LI Shyang – Zhejiang University (China)
- WU Dazhuan – Zheijang University (China)

D3 Case Studies: Aerodynamics II

Cooling of Electrical Motors

- KLEPP Georg – HS-OWL (Germany)
- PRIES Alexander – HS-OWL (Germany)

Effectiveness of Blade Forward Sweep in a Small Industrial Tube-Axial Fan

- MASI Massimo – University of Padova (Italy)
- LAZZARETTO Andrea – University of Padova (Italy)
- CASTEGNARO Stefano – University of Padova (Italy)

Experimental Investigations of Mixed Flow Fans with Various Downstream Angles

- TREDER Anne – Technical University of Berlin (Germany)
- THAMSEN Paul Uwe – Technical University of Berlin (Germany)

E2 Installation Effects

Experimental and Numerical Investigation of Axial Fan Aeroacoustics at Disturbed Inflow Conditions

- LUCIUS Andreas – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- HÜTTER Simon – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- DIETRICH Philipp – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- LEHMANN Marius – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- SCHNEIDER Marc – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- GEYER Thomas – BTU Cottbus (Germany)

Humming Noise Mechanism in Automotive Blower

- HENNER Manuel – Valeo Thermal Systems (France)
- SERAN Aurélien – Valeo Thermal Systems (France)
- DEMORY Bruno – Valeo Thermal Systems (France)
- NAJI Saïd – Valeo Thermal Systems (France)
- CHÉRIAUX Olivier – Valeo Thermal Systems (France)
- AILLOUD Fabrice – Valeo Thermal Systems (France)
- MATHARAN Thibaud – Valeo Thermal System (France)

Experimental Investigation of Installation Effects in a Low-Speed Fan System

- DOMINIQUE Joachim – von Karman Institute (Belgium)
- CHRISTOPHE Julien – von Karman Institute (Belgium)
- SCHRAM Christophe – von Karman Institute (Belgium)

E3 Test Rig & Instrumentation I

Measuring the Flowrate of Fans by Embedded Sensors

- GUÉDEL Alain – CETIAT (France)
- CARÉ Isabelle – CETIAT (France)

Design and Testing of an ISO 5801 Inlet Chamber Test Rig and Related Issues with the Standard

- CASTEGNARO Stefano – University of Padova (Italy)
- MASI Massimo – University of Padova - DTG (Italy)
- LAZZARETTO Andrea – University of Padova - DII (Italy)

Designing, Building and Validating a Test Rig to Measure Two-port Data of Small Fans

- GARNELL Emil – KTH Royal Institute of Technology (Sweden)
- ÅBOM Mats – KTH Royal Institute of Technology (Sweden)
- BANWELL Guy – Dyson (United Kingdom)

F1 Fan Efficiency Metric & Market Surveillance

New Methodologies for Effective Market Surveillance of Large Fans with the INTAS Project

- CHRISTIANSEN Christian Holm – Danish Technological Institute (Denmark)
- WEISS Ingrid – WIP Renewable Energies (Germany)
- JEZDINSKY Tomas – European Copper Institute (Belgium)
- WAIDE Paul – Waide Strategic Efficiency (United Kingdom)
- RUIZ FUENTE Nerea – European Environmental Citizens’ Organisation for Standardisation (Belgium)

A Statistical Survey on the Actual State-of-the-Art Performance of Radial Flow Fans Based on Market Data

- ALDI Nicola – University of Ferrara (Italy)
- CASARI Nicola – University of Ferrara (Italy)
- PINELLI Michele – University of Ferrara (Italy)
- SUMAN Alessio – University of Ferrara (Italy)

Fan Energy Efficiency Metrics and Fan Selection in Commercial and Industrial Fan Systems

- BUBLITZ Mark – The New York Blower Company (United States)
- CATANIA Tom – University of Michigan (United States)
- IVANOVICH Michael – AMCA International (United States)
- MATHSON Tim – Greenheck Fan Corporation (United States)
- MOHSINA Nazme – AMCA International (United States)
- PERSFUL Trinity – Twin City Fan & Blower (United States)
- STEVENS Mark – AMCA International (United States)

The Role of Universities Within the Qualification adn Verification Process of EuP Requirements for Turbomachines using the Example of Rotodynamics Pumps

- LUDWIG Gerhard – Technische Universität Darmstadt (Germany)
- TAUBERT Paul – Technische Universität Darmstadt (Germany)
- PELZ Peter F. – Technische Universität Darmstadt (Germany)

Fan Model Test at Varying Ambient Pressure - Efficient Product Validation at Full Scale Reynolds and Mach Number

- SAUL Sebastian – Technische Universität Darmstadt (Germany)
- PELZ Peter – Technische Universität Darmstadt (Germany)

F2 Noise Prediction by Analytical or Numerical Models II

Comparison of Lighthill's Analogy and Acoustic Perturbation Equations for the Prediction of HVAC Blower Noise

- TAUTZ Matthias – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- KALTENBACHER Manfred – Vienna University of Technology (Austria)
- BECKER Stefan – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

Integrated CFD-Acoustic Approach to the Simulation of Tonal and Broadband Noise Generated by Axial and Centrifugal Fans

- FERRANTE Piergiorgio – NUMECA International (Belgium)
- DI FRANCESCANTONIO Paolo – STS - Scientific and Technical Software (Italy)
- HIRSCH Charles – NUMECA International (Belgium)

Improved Analytical Prediction of Boundary Layer Induced Rotor Noise using Circumferential Modes

- STAGGAT Martin – German Aerospace Center (DLR) (Germany)
- MOREAU Antoine – German Aerospace Center (DLR) (Germany)
- GUÉRIN Sébastien – German Aerospace Center (DLR) (Germany)

Application of a Discontinuous Galerkin Based CAA Solver for Broadband Noise Prediction

- LIBERSON Lev – German Aerospace Center (DLR) (Germany)
- LUMMER Markus – German Aerospace Center (DLR) (Germany)
- MÖSSNER Michael – German Aerospace Center (DLR) (Germany)
- EWERT Roland – German Aerospace Center (DLR) (Germany)
- DELFS Jan Werner – German Aerospace Center (DLR) (Germany)

Evaluation of Analytical Aeroacoustic Models for a Low-Speed Axial Ventilation System

- SANJOSÉ Marlène – Université de Sherbrooke (Canada)
- PESTANA Miguel – Université de Sherbrooke / Ecole Centrale Lyon (Canada)
- MOREAU Stéphane – Université de Sherbrooke (Canada)
- CAULE Patrice – Safran Ventilation Systems (France)

F3a Cordier Guided Design Strategies

Maximal Achievable Efficiency of Fans

- BAMBERGER Konrad – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)

Considerations Concerning the Possible Fan Efficiency in Real Life Applications with Limited Fan Size

- LÖRCHER Frieder – Ziehl-Abegg SE (Germany)
- ANGELIS Walter – Ziehl-Abegg SE (Germany)

F3b Fan Design for Improved Efficiency and Extended Operating Range I

Design Study to Axial Flow Fan Stator Blade Row Used in Reverse Ventilation

- CYRUS Vaclav – AHT Energetika Ltd. (Czech Republic)
- CYRUS Jan – AHT Energetika Ltd. (Czech Republic)

Investigation on the Maximum Total to Static Efficiency of Axial Fans With and Without Diffusers

- WALTER Johannes – Karlsruhe Institute of Technology (Germany)
- CAGLAR Saban – Karlsruhe Institute of Technology (Germany)
- GABI Martin – Karlsruhe Institute of Technology (Germany)

Profiles for Reversible Axial Fans

- KLEPP Georg – HS-OWL (Germany)
- FILIPPI Markus – HS-OWL (Germany)

Friday 20 April 2018

G1 Test Rig & Instrumentation II

A Centrifugal Fan Test Bench for Validation Data at Off-Design Conditions

- PRITZ Balazs – Karlsruhe Institute of Technology (Germany)
- WALTER Johannes – Karlsruhe Institute of Technology (Germany)
- GABI Martin – Karlsruhe Institute of Technology (Germany)

End-of-line Bearing Quality Monitoring on Compact Ventilators - A Case Study

- GRILLIAT Julien – ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
- ANTONI Jérôme – INSA de Lyon (LVA) (France)
- COATS Michael D. – University of New South Wales alumni (Australia)

Integrated Management of Experimental Research- and Meta-Data for Fan Test Rigs

- PREUSS Nils – Technische Universität Darmstadt (Germany)
- PELZ Peter F. – Technische Universität Darmstadt (Germany)

G2 Design for Low-noise Fans I

Investigations on Noise Sources on a Contra-Rotating Axial Fan with Different Modifications

- KRAUSE Ralph – Institut für Luft- und Kältetechnik gGmbH (Germany)
- FRIEBE Christian – Institut für Luft- und Kältetechnik gGmbH (Germany)
- KERSCHER Michael – gfai tech GmbH (Germany)
- PUHLE Christof – GFaI e.V. (Germany)

Influence of Blade Sweep on Aerodynamics and Acoustics of Low-Pressure Axial Fans

- SKORPEL Alexander – Technische Universität Braunschweig (Germany)
- FRANTZHELD Philip – Technische Universität Braunschweig (Germany)
- FRIEDRICHS Jens – Technische Universität Braunschweig (Germany)

Tonal Noise Prediction of Unevenly-Spaced Blades Axial Fans Based on Blade Force Model

- WU Yadong – Shanghai Jiao Tong University (China)
- PAN Dinghao – Shanghai Jiao Tong University (China)
- PENG Zhigang – Shanghai Jiao Tong University (China)
- OUYANG Hua – Shanghai Jiao Tong University (China)

G3 Development of Analytical and Computational Methods

3D Inverse Design Based Optimization of Multi-Blade Row Axial Fans Used for Distributed Propulsion

- ZANGENEH Mehrdad – University College London (United Kingdom)
- WANG Peng – Advanced Design Technology Ltd (United Kingdom)
- GOMES Pedro – Advanced Design Technology Ltd (United Kingdom)

Structure and Kinematics of the Vortex System in Axial Turbomachines

- TAUBERT Paul – Technische Universität Darmstadt (Germany)
- PELZ Peter F. – Technische Universität Darmstadt (Germany)

Extended Scaling Method for Nonsimilarity in Reynolds Number, Stagger Angle and Blade Number

- SAUL Sebastian – Technische Universität Darmstadt (Germany)
- PELZ Peter – Technische Universität Darmstadt (Germany)

H1 Psychoacoustics I

Psychoacoustical Investigation of Sounds from Heat Pumps

- FELDMANN Carolin – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)
- SCHNEIDER Marc – ebm-papst Mulfingen GmbH (Germany)

The Relationship between Perceptual Dimensions of Fan Noise and Patterns of the Specific Loudness

- TÖPKEN Stephan – Carl von Ossietzky University Oldenburg (Germany)
- VAN DE PAR Steven – Carl von Ossietzky University Oldenburg (Germany)

Comparison of Sound Quality Metrics for Axial Flow Fans with Straight and Forward Swept Blades

- MUIYSER Jacques – University of Stellenbosch (South Africa)
- VAN DER SPUY Johan – University of Stellenbosch (South Africa)
- BEKKER Anriëtte – University of Stellenbosch (South Africa)

H2 Design for Low-noise Fans II

Sound Reduction by Leading Edge Serrations in Low-Pressure Axial Fans

- KRÖMER Florian – Friedrich-Alexander University Erlangen-Nürnberg (Germany)
- WESTERMEIER Matthias – Friedrich-Alexander University Erlangen-Nürnberg (Germany)
- RENZ Andreas – Friedrich-Alexander University Erlangen-Nürnberg (Germany)
- BECKER Stefan – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

Influence of the Rotation Speed onto Compact Axial Fans Broadband Noise at Constant Design Point

- GRILLIAT Julien – ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
- BUCHWALD Patrick – University of Stuttgart (ITSM) (Germany)
- VOGT Damian – University of Stuttgart (ITSM) (Germany)
- LAUFER Wolfgang – ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
- SCHNEIDER Marc – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- LUCIUS Andreas – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- SCHMITZ Michael – Siemens AG (Switzerland)

Design of a Low Speed Rim-Supported Fan for Minimum Noise

- HURTADO Mark – Virginia Polytechnic Institute and State University (Virginia Tech) (United States)
- BURDISSO Ricardo – Virginia Polytechnic Institute and State University (Virginia Tech) (United States)

H3 Fan Design for Improved Efficiency and Extended Operating Range II

Numerical and Experimental Investigation of the Velocity Field in Friction Ventilators

- RENZ Andreas – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- PRASS Julian – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- RIEDEL Jörg – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- NADLER Olaf – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
- BECKER Stefan – Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

The Use of Serrated Leading Edge for Inflow Conditioning in Centrifugal Fan

- CARDILLO Lucio – SED Soluzioni Srl (Italy)
- CORSINI Alessandro – Sapienza University of Rome (Italy)
- DELIBRA Giovanni – Sapienza University of Rome (Italy)
- SHEARD Anthony Geoffrey – AGS Consulting Llc (United States)
- TIEGHI Lorenzo – Sapienza University of Rome (Italy)

Fan Unit with Special Guide Vane Design for Low Hub Ratio

- LÖRCHER Frieder – Ziehl-Abegg SE (Germany)
- ANGELIS Walter – Ziehl-Abegg SE (Germany)

J1 Psychoacoustics II

Study on New Parameters for Tonal Noise Evaluation in Small Fan Noise

- MINORIKAWA Gaku – Hosei University (Japan)
- NAKANO Takefumi – Hosei University (Japan)

Acoustic Perception of Irregularly Spaced Blades Fan

- CARNIEL Xavier – CETIM (France)
- DOUCET Alexis – CETIM (France)

Time Structure Analysis of Fan Sounds

- FELDMANN Carolin – University of Siegen (Germany)
- LHONNEUR Joffrey – Ecole Centrale de Nantes (France)
- CAROLUS Thomas – University of Siegen (Germany)
- SCHNEIDER Marc – ebm-papst Mulfingen GmbH & Co. KG (Germany)

J2a Retrofit and Upgrading Existing Fan Installations

Performance Testing of a Retrofitted ACC Fan

- ELS Daniel – University of Stellenbosch (South Africa)
- MUIYSER Jacques – University of Stellenbosch (South Africa)
- VAN DER SPUY Johan – University of Stellenbosch (South Africa)
- MEYER Chris – University of Stellenbosch (South Africa)
- LOUW François – Kelvion Thermal Solutions (South Africa)
- ZAPKE Albert – ENEXIO Management GmbH (Germany)

Energy Saving and Acoustical Optimization - Fan Retrofit for Existing Installations

- HAAG Christian – ebm-papst Mulfingen GmbH & Co. KG (Germany)

J2b Case Studies: Acoustics

Design of Noise Reduced Large Fans for Wind Tunnel Application with CFD-based Optimization - a Case Study

- BAMBERGER Konrad – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)
- KOZUSCHEK Tilo – Howden Axial Fans GmbH (Germany)
- TRYGGESON Henrik – Howden Axial Fans AB (Sweden)

Noise Reduction of a Large Axial Flow Fan for CSP Air-Cooled Condensers

- ANGELINI Gino – Sapienza University of Rome (Italy)
- VOLPONI David – Sapienza University of Rome (Italy)
- WILKINSON Mike – Stellenbosch University (South Africa)
- VAN DER SPUY Sybrand – Stellenbosch University (South Africa)
- BONANNI Tommaso – Sapienza University of Rome (Italy)
- TIEGHI Lorenzo – Sapienza University of Rome (Italy)
- DELIBRA Giovanni – Sapienza University of Rome (Italy)
- CORSINI Alessandro – Sapienza University of Rome (Italy)
- VON BACKSTRÖM Theodor – Stellenbosch University (South Africa)

J3 Automatized Aerodynamic Design via Optimization

Turbo-Machinery Design based on Multi-Physics Fluid-Structure Optimization

- GOKPI Kossivi – INTES FRANCE (France)
- MARCHESINI Jacques – INTES FRANCE (France)
- DEMORY Bruno – Valeo Thermal Systems (France)
- HENNER Manuel – Valeo Thermal Systems (France)

Development, Validation and Application of an Optimization Scheme for Impellers of Centrifugal Fans Using CFD-Trained Metamodels

- BAMBERGER Konrad – University of Siegen (Germany)
- BELZ Julian – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)
- NELLES Oliver – University of Siegen (Germany)

ABSTRACTS

TURBOMACHINERY - PAST AND FUTURE OF CFD SIMULATIONS

Dr. Ing. Georg SCHEUERER ISimQ GmbH (Germany)

The presentation will discuss developments in Computational Fluid Dynamics (CFD) over the last 25 years focusing on turbomachinery and fan flow applications. It will include progress in core technology like turbulence and rotor-stator interaction models as well as algorithmic improvements relating to discretisation schemes and solution algorithms. In addition, advances in critical CFD enablers like geometry modelling, mesh generation, analysis, high-performance and cloud computing and quantification of numerical errors and uncertainties will be reviewed. CFD is not a stand-alone technology anymore. It is increasingly used as a component of multi-physics analysis, the most common fan flow-related examples being fluid-structure and fluid-electromagnetic interactions (motor). These developments will be discussed as part of the presentation. Finally, changes in the usage and user environment of CFD software, from a discretionary validation tool to an obligatory design tool and on to a “digital twin”, and its implications for efficient CFD software packages in terms of performance and user environment will be highlighted.

° ° °

Georg Scheuerer studied Mechanical Engineering at Technische Universität München and did a PhD on turbulence modelling at Universität Karlsruhe. After his dissertation, Georg joined Friedrich-Alexander-Universität Erlangen-Nürnberg where he established and managed a research group on Computational Fluid Dynamics (CFD).

After a short stay at Gesellschaft für Reaktorsicherheit, Georg founded the company Advanced Scientific Computing GmbH which became the predecessor of ANSYS Germany GmbH. At ANSYS Germany, Georg held positions in product development, customer support, consulting and management.

Since 2016, Georg is Managing Director of ISimQ GmbH, a company that specialises on CFD consulting and applications. Georg’s technical background is turbulence, heat transfer, and multiphase flow modelling.

STANDARDS AND REGULATION

Tony BREEN Nuaire (United Kingdom)

Manufacturers produce catalogues that give product details such as size, weight and most importantly performance. Consultants and contractors make decisions based on the data provided. How do they know the data given in the catalogue is correct? Depending on your geographical region, local regulations have been drafted and must be adhered too. These regulations are written by politicians based on advice given from many sources. The aim is to reduce energy consumption and limit the amount of carbon entering the atmosphere. On what basis are these decisions made? In my opinion the answer lies in the application of standards, documents which lay down procedures to determine a fan's performance. What questions will be asked in the future? What happens when the standards don't reflect reality?

° ° °

Tony Breen, is the Technical Support Services Manager at Nuaire, a world leader in the design and manufacture of energy-saving ventilation solutions. A key figure in driving forward the adoption of fan standards both in the UK and internationally, Mr Breen is the Chairman of the ISO and BSI Fan Committees.

Tony started working life as an apprentice with Pressed Steel Fisher, part of British Leyland. After his apprenticeship he gained a BEng (tech) in Mechanical Engineering and M.Sc in Systems engineering at the University of Wales Institute of Science & Technology, in Cardiff. He joined his current employer, Nuaire in 1988, as a test engineer.

During his 30 year tenure at Nuaire, Mr Breen has established the company as a leader in quality and testing of products. As manager of the TSS department, Mr Breen is responsible for new product testing, audit testing and witness testing on behalf of customers.

GOTTHARD TUNNEL VENTILATION SYSTEM DESIGN

Christoph BRANDER Pöyry Schweiz AG (Switzerland)

Since December 2016, the world's longest railway tunnel, the Gotthard Base Tunnel (GBT), built by the AlpTransit Gotthard Ltd, is fully operational. The GBT, having a length of 57 km, a rock oberburden of up to 2300 m, three intermediate construction accesses, rock temperatures of up to 44°C and four emergency stations with smoke extraction, required a complex and robust ventilation system enabling a successful operation of the tunnel under challenging conditions. Along with the different concepts employed for the normal and emergency ventilation, the presentation deals with the major design relevant requirements, the technical implementation and the historical development of the ventilation system of GBT.

° ° °

Christoph Brander, MSc ETH Zurich, works as section head ventilation & mechanics at Pöyry Switzerland Ltd. in Zurich, Switzerland. He studied Mechanical Engineering at the Swiss Federal Institute of Technology (ETH) Zürich, Switzerland. He joined Pöyry 2009, where he held positions as Project Engineer, Senior Engineer and Section Head. His field of specialisation is ventilation, climatic and aerodynamic studies in underground systems and smoke extraction systems. He was responsible for the design and construction supervision of the Gotthard Base Tunnel ventilation systems which contains the main tunnel ventilation and the cross-passage ventilation. He has been accompanying the ventilation system from the project planning and tender phase 2009 to the commissioning in 2015/2016.

FLT Working Group Fans - Benefits and Results of Pre-Competitive Community Research

Erik REICHERT ebm-papst Mulfingen GmbH & Co. KG (Germany)

For more than five decades, the Research Association for Air and Drying Technology (FLT) has been offering its members a platform for pre-competitive industrial community research. Industry experts and scientists from over 20 German research institutes work closely together in project working groupes. Within the working group the FLT members determine and develop the technical topics, objectives and contents of the projects. By participating in the project-accompanying working groupes, the industrial participants benefit from the efficient knowledge transfer with the research centers as well as from the free use of all research results. The close collaboration with the universities also offers the companies further perspectives, such as contact with very well-trained engineering science graduates, as well as the opportunity to initiate bilateral research projects.

At the moment the FLT consists of the two working groups "air conditioning - technical building equipment " and "fans". The fan projects initiated by approximately 20 members of the fan working group - mainly fan manufacturers - are aimed at improving the efficiency and acoustics of axial and radial fans used in industrial applications as well as in room ventilation and air conditioning. Other research areas include the use of new materials, technologies and manufacturing processes as well as the development of innovative concepts for controlling, networking and monitoring the operation of fans.

Since the FLT was established, more than 300 research projects have been initiated. Multi-year projects are funded 100 % by financial resources of the Federal Ministry of Economics and Technology (BMWi) via the Industrial Cooperative Research Associations (AiF). Smaller projects with shorter duration are also financed from the existing resources of the Research Association.

It is intended to provide an overview of FLT's ongoing activities, as well as a review of the projects carried out over the last 15 years by the fan working group. The aim is to summarize which research topics were finished by which universities. Particulary interesting research results are presented and explained. On the one hand a thematic focus is on the investigation of methods and concepts for the aerodynamic optimization of the efficiency and the air performance of fans. Other very interesting challenges are the development of know-how on aeroacoustic source mechanisms and the perception of fan noise as well as the validation of acoustic simulations tools.

ENDLESS FIBRE REINFORCED COMPOSITE-METAL-IMPELLER: MATERIAL RELATED DESIGN AND DIMENSIONING PROCESS FOR HYBRID RADIAL-FANS

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

Due to their excellent mechanical properties, composite materials are becoming increasingly important for energy-related applications. The combination of endless-fibre-reinforced plastic with metallic components allows new hybrid metal-composite-design (MCD), which provides considerable advantages compared to conventional constructions. Innovative approaches of MCD are almost predestined for highly loaded structures like radial impellers, where they show considerable benefits particularly in the optimised use of the materials and the flexible design. Therefore, future concepts can be realised with at least the same functionality but with an increased circumferential speed. In addition to the high efficiency, MCD also enable the possibility of modular concepts which leads to an improvement of the qualification and maintenances processes.

The variety of individual and adjustable material, structural and process parameters result in a complex multi-level design and dimensioning process which impede the practical implementation of radial impellers made of a MCD. The number of the parameters increases enormously, which also leads to numerous possible combinations compared to conventional concepts made of metallic materials. Additional the strong interaction of the disciplines design, dimensioning and manufacturing have to be considered for an efficient design process of hybrid composite structures. Interaction therefore means that the dimensioning process depends on the used manufacturing process. However, the resulting structure of the material yields a related dimensioning and manufacturing process. The manufacturing process also affects the emerging material and influences the possibilities of the construction design, which is classified as an interactive construction and manufacturing process. Metal-material specialized companies will face a new challenge by developing and manufacturing metal-fibre-composite radial impellers for high efficiency applications.

An enhanced interaction of the dimensioning process, with the construction and manufacturing methods and the usage of material based dimensioning methods permits a significant shortening of development times and a reduction of development costs for radial impellers as MCD structures. An example of such a linked product development process for a fibre-composite radial impeller for Technology Readiness Level 1 - 3 is the subject of this paper.

Keywords: Composite Material, Radial Fan, Hybrid Design, Interactive Design Process

INTEGRATIVE SIMULATION METHOD FOR THE PREDICTION OF ANISOTROPIC AND TIME-DEPENDENT MECHANICAL BEHAVIOR OF INJECTION MOLDED FIBER-REINFORCED FAN IMPELLERS – CREEP MODELLING

Sebastian HERTLE Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Andre LASS Universität Rostock (Germany)
Dietmar DRUMMER Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Frank-Hendrik WURM Universität Rostock (Germany)

Injection molded fiber-reinforced plastics are widely used as structural or functional parts in fan applications. Due to the possibility of insert integration in injection molding and the short cycle times without the need of subsequent machining, injection molding is a cost-efficient process for the production of fans. Adding to this is the fact that the high degree of design freedom can be used to achieve lower noise emissions, higher efficiency factors and weight savings. For the latter option, a number of aspects need to be considered. Beneath the oper-ating loads and conditions, flow-induced static and dynamic loads on the surface of the fan can occur and the process-induced anisotropic, time-dependent mechanical behavior needs to be included for long-term use. In order to consider the different aspects, an integrative sim-ulation method is developed. At first a simulation of the flow conditions during the injection process and a flow simulation of the fan is carried out. The results, namely the fiber orienta-tion and pressure profile, are transferred to the structural mechanical model. The mechanical properties of fiber-reinforced plastic parts depend on the fiber orientations caused by the dif-ferent flow conditions during mold filling and the time- and load-dependent creep behavior of the polymer matrix.

Based on experimental approaches the anisotropic and time-dependent creep behavior were investigated and modelled. The Findley power law was used to represent nonlinear viscoelas-tic creep curves and compared to a time-stress superposition principle, which was applied to predict the creep behavior, based on short-term creep curves. The uniaxial tensile creep of a 40 percent by weight glass fiber reinforced injection molded polypropylene was measured into different directions for different specimen thicknesses (1 mm, 2 mm and 4 mm). Creep strength decreased and both creep strain and creep rate increased with loads perpendicular to the injection direction. This effect increases with reduced test specimen thickness and higher loads. Finally, the integrative method is outlined for a fan.

INTEGRATIVE SIMULATION METHOD FOR THE PREDICTION OF ANISOTROPIC AND TIME-DEPENDENT MECHANICAL BEHAVIOR OF INJECTION MOLDED FIBER-REINFORCED FAN IMPELLERS – NUMERICAL AND EXPERIMENTAL APPROACH

Andre LASS University of Rostock (Germany)
Sebastian HERTLE Universty of Erlangen (Germany)
Frank-Hendrik WURM University of Rostock (Germany)
Dietmar DRUMMER Universty of Erlangen (Germany)

Currently installed fans are responsible for more than 10 % of the total European energy consumption. Therefore, minimum efficiencies have been defined to ensure that future developments aim for the lowest energy consumption. Nowadays, fans are increasingly produced using fiber reinforced plastic materials as it contributes to energy and resource savings and reduces weight. Injection molding offers great flexibility in the design of innovative, noise reduced and energy efficient fans, as blade add-ons like winglets, gurney flaps or vortex generators as well as special shaped leading or trailing edges can be incorporated.

In the current effort an integrative simulation method is developed to combine injection molding, aerodynamic and structural mechanic simulations. For this purpose, fiber orientation, material properties and pressure distribution are mapped onto a related structural mechanical model. Thereby, the influence of anisotropic material properties towards the time- and load-dependent displacements of axial and radial fan made of short glass-fiber reinforced plastics can be investigated. The influences of aerodynamic and centrifugal loads were compared to each other in order determine their relevance in the structural design process.

The stress and strain distribution strongly depends on the local fiber orientation of the glass fiber reinforced injection molded material. In order to validate the numerical strain results two strain measurement systems will be presented. First, an optical system based on a three-dimensional and high-resolution digital image correlation was developed to investigate the strain distribution on the surface of an impeller. Second, a strain gauge system was designed, which is capable to measure the local strain within the rotating frame. The second system was specially designed without slip ring contacts for under-water applications and was tested for local strain measurements at a rotor blade under operational conditions. The application of these measurements techniques to axial and radial fans with diameters less than 900 mm is new and allows to obtain up to ten strain voltage signals with a maximum sampling frequency of 1 kHz. Due to its compact design it can be applied to most fans with a hub diameters greater than 110 mm.

POTENTIAL NOISE SOURCE IDENTIFICATION IN A SINGLE STAGE ROTOR-STATOR COMPRESSOR

Peter HARLEY Dyson (United Kingdom)
Raul CORRALEJO Dyson (United Kingdom)
Michael COLLISON Dyson (United Kingdom)
Jimmy LIRVAT Dyson (United Kingdom)
Oscar WILSBY University of Cambridge (United Kingdom)

State legislation for domestic products has driven the need for noise reduction in low Reynolds number turbomachinery. The current study identifies the prevalent noise source type expected in the current design and potential aerodynamic candidates relating to this. A single stage rotor-stator research axial compressor was explored experimentally and numerically. A quantitative acoustic analysis was performed and a qualitative aerodynamic comparison completed to accompany it. The outcome demonstrated the potential aerodynamic flow features that could be related to the specific energy increase seen in the acoustic spectra.

SOUND RADIATION OF A SMOKE-REMOVAL FAN SYSTEM

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

To remove toxic fumes from people caught in fire, firefighters use more and more powerful fan systems. These machines are designed to provide the maximum flow rate when the fire is the most intense. This ventilation power is directly connected with the problem of sound nuisances that interfere with the firefighter communications even if helmets are worn. The noise also causes premature fatigue for the firefighters. On a more long term, this sound nuisance (mainly the high-pitched tonal noise at the blade passing frequency and its harmonics) can trigger losses of hearing and eventually deafness, or other health hazards for the firefighters (increased heart diseases for instance). The present study is meant to provide details characterizations of the radiated sound of such systems and identify ways of reducing the associated noise.

A representative fire fan system has then been studied in the recently-built anechoic wind tunnel at Université de Sherbrooke at several operating conditions. It consists of a thermal gas engine connected to a fan. The engine is a 6 Hp Honda running from about 1440 RPM at idle to 3660 RPM at full speed. The fan itself is protected by a front and a rear grill, both attached to a volute. The fan tip clearance is more than 1 cm for safety purpose. The size of the fan is 21 inches in diameter. The fan has 7 blades. It can be tilted from 0 to 20 degrees to blow the air slightly upward. Far-field sound pressure spectra and directivity have been obtained. Three different configurations have been studied and compared: the complete fan system, the fan system without the front grill, and the fan without front grill and volute. The microphone directivity array consists of 19 Bruel and Kjaer (B&K) microphones. Excellent repeatability is obtained on the broadband noise (maximum variation of 0.5 dB) and a good one on the tonal noise (maximum variations of 2-3 dB). Scaling laws of the power spectral densities of the far-field acoustic pressure with fan speed have been deduced. Moreover noise source localization has been achieved with two different methods. Conventional beamforming technique is compared with a new de-dopplerization technique for rotating sources. The source localization array consists of a 60 microphone logarithmic spiral array.

Finally having shown the main dipolar noise feature of sound of the fire fan system and the respective contribution of the thermal engine and the fan, a method to separate rotating and stationary noise sources has been successfully applied. The contribution of both noise components are then clearly identified, which allows a separate analysis of both noise sources. Conclusions are finally drawn and suggestions for future noise control are provided.

INVESTIGATION ON THE NOISE OF AN AXIAL LOW MACH-NUMBER STAGE WITH A HETEROGENEOUS STATOR

Miguel PESTANA	Safran Aircraft Engines, École Centrale de Lyon, Université de Sherbrooke (France)
Marlène SANJOSÉ	Université de Sherbrooke (Canada)
Stéphane MOREAU	Université de Sherbrooke (Canada)
Michel ROGER	École Centrale de Lyon (France)
Mathieu GRUBER	Safran Aircraft Engines (France)

A low-Mach number ducted axial fan from an aircraft air-conditioning system is investigated in this study. This configuration has a short rotor-stator distance and a heterogeneous (non-periodic) stator row. Those features strongly modify the rotor-stator noise-generation mechanisms. Two numerical simulations with the original heterogeneous and a homogeneous OGV are performed using the Lattice-Boltzmann Method (LBM). A comprehensive acoustic analysis is made upstream of the fan in order to retrieve the spectral and modal content of the noise. The direct comparison of the homogeneous and heterogeneous configurations provides the quantification of the impact of a heterogeneous stator. Finally, the dominant modes caused by the stator heterogeneity are identified by comparing with an experiment.

TIP LEAKAGE FLOW AND ITS IMPLICATION ON THE ACOUSTIC SIGNATURE OF A LOW-SPEED FAN

Aurélien MARSAN Université de Sherbrooke (Canada)
Lallier-Daniels DOMINIC Université de Sherbrooke (Canada)
Marlène SANJOSÉ Université de Sherbrooke (Canada)
Stéphane MOREAU Université de Sherbrooke (Canada)

Fans and blowers are omnipresent in daily life. They are utilized in large scale Heating, Ventilation and Air-Conditioning (HVAC) systems, automotive cooling systems as well as small scale systems such as electronic cooling systems for home computers. They represent as many sources of noise disturbances.

Meanwhile, the health impacts of environmental noise are fast becoming a growing concern. A recent report from the World Health Organization concludes that ”If negative effects on sleep are to be avoided the equivalent sound pressure level should not exceed 30 dBA indoors for continuous noise”. This is a challenge for fan and blower manufacturers. A new generation of both efficient and quiet air moving systems is needed. For that purpose, a better understanding of the noise generation mechanisms in low-speed fans and blowers is required.

Within this scope, a consortium named Ultra High-Efficiency Quiet (UHEQ) Fans has been funded between Michigan State University (MSU), University of Notre Dame (ND) and Université de Sherbrooke (UdeS). The first phase of this consortium, from 2010 to 2015, focused on the collection of experimental data sets. The present phase of the UHEQ consortium aims at using advanced numerical tools together with the available experimental database in order to give a strong understanding of the noise generation mechanisms in blowers and fans.

The fan investigated in this study, dubbed Rotating Controlled Diffusion Blade (RCDB), is a research fan that has been subject to an extensive experimental investigation using the Axial Fan Research and Development (AFRD) test facility at MSU. The AFRD facility consists in a large plenum of 3x3x2 m large, the test fan being mounted in an opening on the ceiling wall. The fan has a diameter of 740 mm, and a 3 mm annular tip clearance with the shroud surface; the blade span is 135.2 mm and the controlled diffusion (CD) profiles forming the blades have a constant chord of 133.9 mm along the span. The RCDB was designed to be modular and is able to accommodate anywhere from 2 to 9 blades. The design flow rate of the fan is 2.31 kg/s and the design rotational speed is 437 RPM. Available experimental data from the AFRD facility includes wall-pressure measurements on the fan blades along the chord for several radial positions. Hot-wire anemometry and PIV measurements were also conducted to provide an in-depth description of the flowfield encountered downstream of the fan. More recently, aeroacoustic measurements were also carried out within the non-anechoic AFRD facility.

For the purposes of this study, the 3-blade RCDB configuration within the full AFRD facility was simulated using a lattice-Boltzmann method (LBM) code to further investigate the entire flowfield around the fan. Three operating points covering the entire stable operating range are covered and a detailed aeroacoustic analysis is carried out in order to especially investigate the role of tip leakage flow on the acoustic footprint. A detailed analysis of the trajectory and of the unsteady behaviour of the tip-leakage flow is performed. The main flow features and their connections with typical acoustic signatures are explained.

DESIGN AND INVESTIGATION OF A MULTISTAGE AXIAL CONTRA-ROTATING FAN

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

Axial blowing fans are common in many fields of application. By using an axial fan it should be noted there is a swirl occurring at the trailing edge of the blades due to the working principle. The swirl is not needed in most cases and may have unfavourable influence on subsequent devices, e.g. higher pressure drop or lower heat transfer coefficient. As the static pressure rise is an evaluation criterion for the fan efficiency there are different possibilities for converting the dynamic pressure of the swirl into a static pressure rise. The most common application for rising the efficiency is the installation of discharge guide vanes.

Contra-rotating axial fans (CRF) are well known as one opportunity to increase the efficiency of a fan, too. The efficiency increase is driven by the conversion of the dynamic pressure of the swirl at the trailing edge of the first axial fan blades into a static pressure at the end of the whole stage. In comparison with fans using discharge guide vanes, a higher power density can be obtained.

The development of a multistage axial fan using contra-rotating wheels at each stage is described here. The outer diameter of the fan is 120 mm and the rotational speed for the first and the second fan is are 133.3 s-1 and 83.3 s-1, respectively. The design of the complete fan based upon the design approach for contra-rotating axial fans of CFturbo. The first impeller was designed for a performance of 2/3 of the overall power. Hence, the balancing of momentum was not fulfilled although that is an available design option in CFturbo amongst others.

The contra-rotating axial fan is examined with different methods. Numerical calculations as well as performance measurements and measurements on the velocity profile with optical methods (PIV) are provided to understand the working principle of every stage. The high efficiency and the axial outlet flow without tangential velocity components are demonstrated with these measurements.

PARAMETRIC STUDY OF VOLUTES FOR OPTIMAL CENTRIFUGAL FAN IMPELLERS

Ardit GJETA Polytechnic University of Tirana (Albania)
Konrad BAMBERGER University of Siegen (Germany)
Thomas CAROLUS University of Siegen (Germany)
Andonaq LONDO Polytechnic University of Tirana (Albania)

European regulations set standards for energy-efficiency of fans, with increasing requirements in the near future. Many studies concerning centrifugal fans have investigated the impeller but only to a smaller extends the spiral casing (volute). The volute may take up a substantial part of the fan's hydraulic loss. Hence, appropriate design of the fan volute has significant meaning to centrifugal fan performance.

Following a current and nearly finished study on aerodynamic optimization of centrifugal fan impellers using CFD-trained meta-models, where a method for optimization of impellers of the whole class of centrifugal fans has been developed, an extension to the volute is envisaged. For that, a parametric study of volutes is carried out which should lead to advanced best practice recommendations for the volute design.

Starting points are optimal impellers for the whole range of specific speeds. The detailed flow field at the impeller's outlet from preceding RANS simulations is used as boundary conditions for a RANS of the flow in the volutes. An automized loop with RANS and data post-processing is set up for allowing a large number of parameter variations.

The effect of volute angle, volute width (as a fraction of impeller exit width), geometrical parameters related to the tongue and axial position of the rotor on total pressure loss and static pressure recovery coefficient are presented. Selected designs are investigated experimentally for a preliminary validation.

Blade Design and Optimization of Low Pressure Axial Fans using a Cascade Approach

Heiko WINTERBERG Volkswagen AG (Germany)
Jens FRIEDRICHS Technische Universität Braunschweig (Germany)

Background: Low to intermediate solidity axial fans (0.4 ≤ sigma ≤ 1.2) can often be found in heating, ventilation and air conditioning (HVAC) systems. In many applications (e.g. automotive cooling fans) the aerodynamic-design aims for high flow rates at low pressure rises. Therefore those fans usually have no stator, a low hub-to-tip ratio (≤ 0.4) and few blades.

While cascade based design strategies are by definition used for high solidities (sigma ≥ 1), the isolated airfoil approach is valid for low solidities only (sigma ≤ 0.7). For medium solidity fans the literature recommends mostly the isolated airfoil approach with additional empirical corrections to take interference-effects into account. With the aim of using a cascade based fan-design for low to medium solidities, a new design methodology with suitable cascade-data was developed.

Used Methods: For the purpose of low to medium solidity fan-blade-design a MATLAB based program was developed. This tool consists of three main classes and generates the three-dimensional fan geometry in the following manner:

1. Prediction of the velocity triangles at in- and outlet according to the required operating point using a general fixed vortex approach
2. Determination of best efficiency airfoil data (e.g. deviation and incidence angles) based on cascade simulations
3. Generation and stacking of two-dimensional blade-sections allowing for a flexible incorporation of blade sweep
4. Aerodynamic load corrections to take blade-sweep into account

The efficiency of this initial design is further improved by doing 3D-CFD based optimizations using OpenFOAM. Therefore an optimization procedure with nine geometric parameters is used.

Results: Comparing the predicted pressure rise and the velocity triangles with 3D-CFD data, a good agreement can be found. Together with the optimization procedure a fast and reliable design process for low-to-medium solidity fans was developed. The paper shows the design principles using the example of an automotive cooling fan to get reliable initial designs in minutes. Furthermore the optimization strategy is described and results are discussed.

FLOW LOSS AND FLOW STRUCTURE OF CIRCULAR ARC BLADES WITH DIFFERENT LEADING AND TRAILING EDGES

Tao BIAN Jianghan University (China)
Christian LANDFESTER Technical University Kaiserslautern (Germany)
Jun FENG Jianghan University (China)
Qianpeng HAN Jianghan University (China)
Martin BÖHLE Technical University Kaiserslautern (Germany)

Compared with circular arc blade with rounded leading and trailing edges, the circular arc blade with flat leading and trailing edges can be produced relatively cheaper by NC-milling machine. However, the flat leading and trailing edge of the blade may also cause relatively higher flow loss. At present, there is few available information in the open literature focusing on the flow loss and behavior of circular arc blade. In particular, the influence of leading and trailing edge geometry on the flow structure remains unknown. Therefore, two questions are raised: Firstly, how large is the flow loss and flow turning angle of circular arc blade. Secondly, how much is the influence of the leading and trailing edge geometry on flow loss and flow structure.

In this paper two kinds of circular arc blade with constant thickness are examined by numerical and experimental methods. The first circular arc blade has rounded leading and trailing edge, the leading and trailing edge of the second blade are flat. Both blades have a chord length of 100 mm and a camber angel φ of φ = 20 degrees. The stagger angle λ = 30 degrees. The investigated spacing ratio t/l is 1.0. In order to investigate the influence of Reynolds number on the flow loss and flow structure, the Reynolds Number was Re = 200000 and Re = 400000. All examinations were performed for different incidence angles from i = -15 degrees to i = 10 degrees with a stepwise movement of 5 degrees.

The paper shows flow loss in dependence of incidence and Reynolds number. The influence of the sidewall on the flow loss and flow structure is also taken into account. The difference between two- and three-dimensional flow loss of both investigated blades is concluded. The flow loss and flow structure of both blades are investigated and compared. The flow structure is shown on the basis of numerical and experimental oil-flow picture. Especially, the flow behavior such as separation bubble at the leading edge, secondary flow and the flow structure in the corner between the blade and the corresponding sidewall is shown and discussed.

Analytical Approach for the Performance Prediction of High Pressure Centrifugal Fans in Serial Arrangements

Martin NUDISCHER University of Stuttgart (Germany)
Hansgeorg BINZ University of Stuttgart (Germany)
Matthias BACHMANN University of Stuttgart (Germany)
Stefan RECKER University of Stuttgart (Germany)

Centrifugal fans are commonly used as a component in industrial processes. The performance of the installed fan is often adapted to one defined operating point. By a change of the system resistances there is also a change in the operation point of the fan. Especially increased resistances, caused by an extension of the pipe system for example, can significantly shift the operating point of the fan to low efficiency areas up to the point of insufficient output. To meet these new conditions there are several options to increase the fan capacity. The first option to meet the higher demand is to accelerate the fan impeller which results in a higher operating speed, until the maximum permissible strength is achieved. If this option is not sufficient, the fan is often replaced by a larger fan. Another option to increase the available pressure is to combine two or more fans in a serial arrangement.

Despite the advantages of higher pressure, efficiency benefits at variable operating points as well as more variability in structural arrangement and electrical requirements, only theoretical descriptions to reliably design serially arranged fans are given in the relevant literature of fans. The most common method to calculate the performance of serially arranged fans is to sum up the pressure increases of each single stage at a constant volume flow rate. However, this method indicates deviations compared to measured results. In previous investigations it could be shown, that the first stage in a serial arrangement has a behavior comparable to the single fan. Despite the geometrically identical structure of the used fans, the second stage showed a different behavior. Hence, there is a need for the manufacturer to reliably predict the performance of serially arranged fans based on the data of the single fan. This paper introduces a new method to calculate the performance based on the specific energy. The method is evaluated with measurement results of different fans. The introduced method covers serial arrangements of identical fans as well as serially arranged combinations of different fans.

Malfunctioning of Air-Delivering Systems: Examples of Fan Driven Transient and Oscillating Flows

Thomas CAROLUS University of Siegen (Germany)
Bernd HOMRIGHAUSEN University of Siegen (Germany)

The paper reports the analysis of two initially malfunctioning industrial air-delivering systems. In both cases the air-delivering fans were the drivers for the undesired performance - not because the fans were per se of poor design but the interplay with the particular system created an unforeseen transient or oscillating flow.

The first example deals with a huge silo for powdered goods in the food industry. The powder is conveyed into the silo via an aerating and fluidizing air stream. During the first start up the silo collapsed. A network analysis of the air path showed that the dimensioning of the air supplying screw compressor, safety valve and exhaust air fan was inappropriate - the comparably small exhaust fan was able to create the damaging negative gauge pressure inside the silo at a certain time instance during the loading of the silo.

The second example is a large central air conditioning system for a clean room laboratory building consisting of two parallel supply and two parallel exhaust air handing units. The pressure in the supply air handling units showed intolerably high amplitude pressure oscillation in the order of 10 Hz with the consequence that the walls of the unit were moving visibly. From measurements in a series of systematic experiments a number of possible known mechanisms were checked: vortex-generated unsteady flow phenomena and standing wave resonance phenomena, interaction of the two parallel supply and/or exhaust air handling units and self-excited surge of the system due operation of the fan on its characteristic with a positive volume flow rate/static pressure rise slope. A possible correlation of frequency and amplitude of the pressure oscillations with the mean flow velocity and the blade passing frequency of the rotor was checked by varying the fan speed. By increasing and reducing provisionally the pressure resistance in the flow path one of the fan's actual characteristic and its operating point were determined experimentally. Moreover, the unsteady volume flow rate through each flow of one of the double-flow fans in one of the supply units was determined. Here, an alternating suction of the two flows of one of the fans was identified which is believed to be a rather unique mechanism. It could be weakened by rectifying grids in the inlet of both flows, with the consequence that the overall pressure oscillations were reduced substantially.

TECHNICAL OPERATIONS RESEARCH (TOR) - ALGORITHMS, NOT ENGINEERS, DESIGN OPTIMAL ENERGY EFFICIENT AND RESILIENT COOLING SYSTEMS

Philipp LEISE Technische Universität Darmstadt (Germany)
Lena ALTHERR Technische Universität Darmstadt (Germany)
Peter PELZ Technische Universität Darmstadt (Germany)

The overall energy efficiency of ventilation systems can be improved by considering not only single components, but by considering as well the interplay between every part of the system. With the help of the method "TOR" ("Technical Operations Research"), which was developed at the Chair of Fluid Systems at TU Darmstadt, it is possible to improve the energy efficiency of the whole system by considering all possible design choices programmatically. We show the ability of this systematic design approach with a ventilation system for buildings as a use case example.

Based on a Mixed-Integer Nonlinear Program (MINLP) we model the ventilation system. We use binary variables to model the selection of different pipe diameters. Multiple fans are model with the help of scaling laws. The whole system is represented by a graph, where the edges represent the pipes and fans and the nodes represents the source of air for cooling and the sinks, that have to be cooled. At the beginning, the human designer chooses a construction kit of different suitable fans and pipes of different diameters and different load cases. These boundary conditions define a variety of different possible system topologies. It is not possible to consider all topologies by hand. With the help of state of the art solvers, on the other side, it is possible to solve this MINLP.

Next to this, we also consider the effects of malfunctions in different components. Therefore, we show a first approach to measure the resilience of the shown example use case. Further, we compare the conventional approach with designs that are more resilient. These more resilient designs are derived by extending the before mentioned model with further constraints, that consider explicitly the resilience of the overall system. We show that it is possible to design resilient systems with this method already in the early design stage and compare the energy efficiency and resilience of these different system designs.

SOFTSENSOR FOR THE CHARACTERISATION OF THE PROCESS FLUID

Christian SCHMITZ Technische Universität Darmstadt - Industrial Science GmbH (Germany)
Mehdi NAKHJIRI Industrial Science GmbH (Germany)
Peter PELZ Technische Universität Darmstadt (Germany)

The main function of fans is supplying the required volumetric flow of process gas. Among others for or from combustion processes, for chemical processes or for energy transport. The knowledge of the gas composition, i.e. its characteristic properties, is an important information for process control as uncertainty in gas composition is often accompanied by an uncertainty in the process mass flow, which may even lead to critical operation points and early failure. For diagnosis applications, e.g. for leakage detection, the knowledge of the gas composition is as well essential.

The paper introduces a method to substitute expensive volume flow and gas composition sensors using process knowledge and available data from cheap and often already existing sensors. It is shown that for a two-phase fluid, the characteristic map of the fan and the mixture laws of ideal gases contain enough information to calculate the volume flow rate and the gas composition if the inlet gas temperature, the pressure rise over the fan and the fan converter data, i.e. rotational speed and electrical power input, are known.

The concept is validated experimentally. The used test rig consists of a temperature sensor, the radial fan, a pressure sensor, a throttle valve and a volume flow sensor. Validation was carried out by comparing the data of the latter as well as the used gas composition with the results of the softsensor.

EXTENDED RANKINE-BETZ THEORY FOR DESIGN OF TUNNEL VENTILATION SYSTEMS

Peter PELZ Technische Universität Darmstadt (Germany)
Sebastian SAUL Technische Universität Darmstadt (Germany)
Christopher DOBERSTEIN Technische Universität Darmstadt (Germany)

Today the system design of tunnel ventilation systems including jet fans neglects the stream tube contraction being covered by the Rankine-Betz theory. Thus, the presented system design tool extends in a truly physical manner the current method.

The influence of the stream tube contraction is discussed and compared to the design method. In addition, the traffic is treated as a peristaltic flow at high Reynolds number and friction losses are modeled with common approaches. The new model allows analysis of several extreme situations, like normal traffic flow, traffic jam and fire in a tunnel.

The results of the design tool are suitable velocity ratios of fan and tunnel as well as power consumption and operation condition.

INVESTIGATION OF DIFFERENT FAN NOISE PREDICTION METHODS

Clemens JUNGER Technische Universität Wien (Austria)
Manfred KALTENBACHER Technische Universität Wien (Austria)

Fan noise prediction is a necessary stage in all modern fan design processes. For the noise prediction a lot of different methods exist, which can be roughly categorized in three different classes. The first class contains empirical methods, based on geometry and operating conditions. They are very fast and easy to apply. The second class contains semi-empirical methods that are partly based on flow quantities from measurements or simulations. These methods shall represent the actual operating conditions in a better way, but are more time consuming since the flow quantities must be obtained. The third class contains methods that compute the noise radiation directly from fluctuating quantities of the transient flow. These methods give a good insight in the contribution to the overall sound level from different sound sources. Due to the need of transient flow data, this class of methods is highly demanding in computational power.

In our contribution we compare different methods from all three classes and apply them to a low-pressure axial fan. The fan used for the application is a previously published benchmark case. The generic fan is a typical fan to be used in commercial applications. The rotor benchmark fan provides an extensive amount of measurement data including aerodynamic performance (volume flow rate, pressure rise and efficiency), wall pressure fluctuations in the tip gap region, fluid mechanical quantities on the fan suction and pressure side (velocity in three spatial direction and turbulent kinetic energy) and acoustic spectra at different microphone positions. Our numerical approach from class three is based on a forward coupling between a flow simulation with ANSYS Fluent and an aeroacoustic source term and wave propagation computation with multiphysics research software CFS++. The comparison of the different methods and the measurement include averaged sound power level as well as spectral sound power level.

NUMERICAL ANALYSIS OF THE ACOUSTIC FIELD OF AN AXIAL FAN AT DISTURBED AND UNDISTURBED INFLOW CONDITIONS

Seyed Mohsen ALAVI MOGHADAM RWTH Aachen University (Germany)
Alexej POGORELOV RWTH Aachen University (Germany)
Benedikt ROIDL RWTH Aachen University (Germany)
Matthias MEINKE RWTH Aachen University (Germany)
Wolfgang SCHRÖDER RWTH Aachen University (Germany)

The flow in the tip region of axial fans plays a major role in the aerodynamic losses, thermal loads and acoustic emissions. In this paper the flow and the acoustic field of an axial fan will be analyzed by a hybrid CFD/CAA method.

In a first step, the tip clearance flow in a ducted axial fan is predicted by LES. The simulations are based on a multi-block structured mesh with 140 million grid points for a single blade which comprises a 72 degree segment of a rotating axial fan with periodic boundary conditions in the circumferential direction. The turbulent flow is simulated at a Reynolds number of 936000 operating at a design and off-design condition for various tip gap sizes in order to determine the acoustic sources. Computations are performed with undisturbed inflow as well as disturbed inflow conditions using a synthetic turbulence generation method (Roidl, B. et al. Int. J. Heat Fluid Flow 2013). Comparisons of the simulation results with the experimental data (Tao, Z. and Carolus T.H. ASME 2013) including spectra of the surface pressure fluctuations show good agreement.

In off-design operating condition the tip clearance vortex interacts with the downstream blade generating a cyclic transition with distinct interaction frequency on the suction side of the blade near the tip. The turbulent interaction at the inlet dissipates the tip leakage vortex and triggers transition on a larger surface area of the blade. The acoustic field on the near field and the far-field is determined by solving the Acoustic Perturbation Equations (APE) on a mesh for a single blade consisting of approx. 1.1 billion grid points. The maximum resolvable frequency achieved by the computational mesh is about 10 kHz. The acoustic source terms are determined from the flow field results. The acoustic results in terms of the noise spectra are in a good agreement with experimental data and show an increasing tip gap size to lead a higher broadband noise level. More detailed results with turbulent inflow conditions and acoustic source localization will be presented in the conference paper.

NUMERICAL AEROACOUSTIC ANALYSIS OF SMALL COOLING FANS FOR ELECTRONIC DEVICES

Alexis TALBOT Free Field Technologies (Belgium)
Yves DETANDT Free Field Technologies (Belgium)
Shogo HATANAKA Software Cradle Co., Ltd (Japan)
Kenichiro HIRAI Software Cradle Co., Ltd (Japan)
Gaku MINARIKAWA Hosei University (Japan)

Most electronic devices require efficient cooling systems in order for them to operate correctly. Fans often take an important place in this cooling system, extracting the heat outside the casing. As these electronic components are present in quiet environments (home, office, conference rooms) and acoustic performances represent a competitive advantage for fan suppliers, the cooling systems in general and fans in particular need to be as quiet as possible.

The aerodynamic and thermal performances are predicted by means of computational fluid dynamics (CFD) simulations. To name but a few, the number of blades, the blade shape, and the rotation speed are parameters which are optimized. These parameters will also have an influence on the acoustic signature of the fan. Some dominant tonal components may be critical for instance and acoustic simulation results can be integrated in this optimization.

In this paper, we compare the noise generated by cooling fans which have the same size, but which differ by their parameters (number of blades, blade profile, ...). A hybrid technique is used for this analysis, the unsteady turbulent field is computed in a first step, and once the acoustic sources are extracted by processing the turbulent field, the acoustics is propagated in the acoustic domain, up to virtual microphones. The simulation process is presented in details, highlighting some critical parameters and their influence on the numerical accuracy. The results are illustrated with maps showing the acoustic propagation mechanisms, and the regions where the most important sources are generated.

The comparison is performed in free field conditions for validation purpose, but the method could be applied similarly in an installed configuration. The numerical results are compared to experimental measurements performed in anechoic facility. The paper reports the numerical performances of the computational process and the CFD and acoustic models are presented. The influence of some parameters (such as the turbulence level) on the acoustic results is discussed. The results are compared to the experimental measurements for the different fans.

OBSERVATION OF NOISE OF FAN WITH OBSTACLE FOR ELECTRIC COMPONENTS

Kimihisa KANEKO Fuji Electric Co., Ltd. (Japan)
Satoshi MATSUMOTO Fuji Electric Co., Ltd. (Japan)
Tsutomu YAMAMOTO Fuji Electric Co., Ltd. (Japan)

The recent trend in downsizing electric equipment has caused an increase in the heat generation density and an accompanying need to increase air flow to enable cooling. Therefore, the aerodynamic noise from the fan is the dominant source of noise in air-cooled electrical equipment. Fan noise is known to be influenced by the fan operating point and the fan's surrounding structures.

This study presents the use of experimental and numerical simulations to observe the noise made by a fan used to cool electrical components when interacting with an obstacle.

A simplified experimental setup was designed consisting of three parts: a fan, a duct, and an obstacle. The one-block obstacle in the experimental model was assumed to be an aggregate electric component. The sound pressure level (SPL) was found to be influenced by the distance between the obstacle and the fan. Decreasing these distances, the SPL are increasing, particularly below a half distance of the fan diameter, SPL is drastically increased.

To conduct a numerical analysis of aerodynamics noise, either the hybrid computational aeroacoustics (CAA) method or the direct CAA method can be employed. The hybrid CAA method can be assumed to separately solve incompressible turbulent flow and the acoustic wave. The direct CAA method uses compressible Navier-Stokes equations to compute both the turbulent flow and the acoustic wave. In our study, the direct CAA method is used to consider the aero acoustic interactions between the fan and the obstacle under a low Mach number flow. A compressible Large Eddy Simulation (LES) approach was used to predict the SPL. The numerical simulation captured the sound pressure propagation toward the sound observation point. And it was found that the SPL influenced by the distance between the fan and the obstacle can also captured. This method was found to be applicable for predicting the SPL when developing low-noise products.

UNSTEADY SIMULATION OF TONAL NOISE FROM ISOLATED CENTRIFUGAL FAN

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

In this study, tonal noise from an isolated centrifugal fan is investigated using unsteady Reynolds-averaged Navier-Stokes (URANS) simulations. Isolated centrifugal fans are often used in ventilation systems and play an important role in producing tonal and broadband noise. The broadband noise can be reduced when the fan efficiency is optimized. However, the tonal noise cannot be effectively reduced. It is therefore of great interest to identify and reduce the tonal noise for this type of fans operating in public environment. The noise is predicted by coupling the URANS and the Ffowcs Williams and Hawkings acoustic analogy. The numerical methodology and mesh generation methods are validated. Numerical simulation is a more convenient way to identify tonal noise than experiments. However, simulation of the noise using advanced computational fluid dynamics (CFD) methods (e.g., large eddy simulation) requires many computational resources. To predict the tonal noise, a potentially convenient method is the URANS. The method can simulate characteristic unsteady structures, which are responsible for the tonal noise generation, with low computational costs. Though it has a drawback to provide the fluctuations that are important for the broadband noise generation.

The aerodynamic properties obtained from RANS and URANS are consistent with the experimental data. The magnitudes of the tonal noise at the blade passing frequencies are well predicted. Moreover, the broadband noise below 350 Hz agrees with the measurement although obvious discrepancies are found at high frequencies, which cannot be resolved by URANS. Recirculating flows, which are responsible for reducing the fan efficiency and increasing the noise generation, are observed between the shroud and the blade trailing edges. It is found that the recirculating flows are associated with the gap that is between the shroud and the inlet duct.

DEVELOPMENT OF HIGH EFFICIENCY FAN SYSTEM FOR OUTDOOR UNIT OF AIR-CONDITIONER

Taku IWASE	Hitachi, Ltd., R&D Group (Japan)
Erika KATAYAMA	Hitachi, Ltd., R&D Group (Japan)
Sachio SEKIYA	Hitachi, Ltd., R&D Group (Japan)
Tetsushi KISHITANI	Hitachi-Johnson Controls Air Conditioning, Inc. (Japan)

Developing high efficiency air-conditioners is currently a pressing demand for environmental conservation and energy saving. Fan power consumption comprises a large percentage of the total energy consumption of an air-conditioner. High efficiency fan systems therefore directly contribute to energy saving of the air-conditioner.

Recently, it is important to consider influence of equipped condition in the development of fan system. We therefore estimated flow rate and fan shaft power considering the condition equipped with the fan systems in the air-conditioner by computational fluid dynamics (CFD). The feature of the estimation is that we can consider an influence of components which are propeller, bell mouth, motor and heat exchanger etc. The computational model consists of doom shaped region, region of the outdoor unit and rotating region around propeller. The grids were composed of tetra and prism elements. The prism elements are set on blade surface. Part around propeller was in the rotating frame of reference. The other parts were in the stationary frames. The numbers of elements were approximately from 25 million to 35 million. The numerical simulation code employed an incompressible Reynolds-averaged Navier-Stokes simulation (RANS) commercial solver. It also employed a k-epsilon turbulence model and wall function. Heat exchanger was simulated by porous model. Moreover, we analyze total pressure loss in the outdoor unit by investigating calculated results.

It was confirmed by CFD that the outlet loss of fan was the largest. The inner loss of fan was second and the loss of heat exchanger was third. From this analysis, we could propose the new fan system of 2 fan type, improved long bell mouth, and sigma shaped heat exchanger. We confirmed by CFD that the fan shaft power of the developed outdoor unit decreased by 56 % compared to the old outdoor unit. The effect of the improvement of the bell mouth was 22 %. That of the increase of number of fan was also 22 %. That of the improvement of heat exchanger shape was 12 %. Finally, we checked the high efficiency effects experimentally. As a result, we confirmed that the fan shaft power of the developed outdoor unit decreased by 50 %.

INFLUENCE OF THE ENCLOSURE ON THE PERFORMANCE OF RADIAL FANS

Georg KLEPP OWL University of Applied Science (Germany)
Walter KAMPHAUSEN OWL University of Applied Science (Germany)

Due to restrictions in the space that is available for mounting fans, a fan sometimes has to work in a narrowed enclosure. This leads to reductions in the fan performance and an increase in fan noise. A common arrangement is a radial impeller confined in a square housing. The flow patterns in the enclosure and their influence on the fan performance are investigated. For this investigation the complex three dimensional transient flow is described using a simplified two dimensional stationary model, reducing the complexity of the problem to the main design parameters. The analysis is performed using Computational Fluid Dynamics (CFD).

Different sizes of the enclosure and different impeller locations are investigated as well as different types of impeller (slow runner and fast runner). As the enclosure narrows, the performance mostly decreases. For certain configurations the enclosure leads to a pressure increase at the the outlet. If the enclosure is too big, the influence on the fan performance is negligible. If the enclosure is too small, the performance is strongly diminished and the influence of other design parameters is negligible. The pressure at the outlet and thus the fan performance is strongly influenced by the flow patterns in the enclosure. The fan performance is mainly affected by the maldistribution of the air at the outlet, the flow and losses in the confinement and the influence of the non-uniform pressure distribution at the impeller outlet. These effects depend on the size and dimensions of the enclosure and the impeller diameter, as well as the location of the impeller and flow patterns at the impeller outlet.

A correlation of the performance reduction with the geometric parameters (dimensions of the enclosure and fan location relative to impeller diameter) and flow pattern at the impeller outlet is attempted. Using the results some indications for an optimization of the performance of radial impellers in square enclosures are derived.

A STUDY ON DESIGN OPTIMIZATION OF CENTRIFUGAL FAN OF HIGH VOLTAGE GENERATOR BY NUMERICAL ANALYSIS

Chungman JANG Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)
Joonyeob LEE Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)
Moojong JEON Hyundai Electric & Energy Systems Co., LTD. (Republic of Korea)

A generator is a device that converts the mechanical energy provided by an engine, a gas turbine, a steam turbine, etc. to electric energy. Heat is generated during the energy conversion process and the internal temperature of the generator is increased. A cooling fan is used to suppress the generator temperature characteristic within the numerical value in the standard. Choosing a cooling fan in a generator cooling design is a key core technology. The required air volume and required static pressure must be satisfied and the influence of the mechanical loss and noise must be considered. If the fan selection is not appropriate, it will affect the generator performance and increase the. Therefore, an effective cooling fan design is important for the selection of an optimal cooling fan.

The high voltage generator consists of a rotor, a stator, a winding, a cooling fan, and a heat exchanger. The internal structure is complicated, and the flow resistance is generated as the cooling channel is either enlarged or reduced. Flow resistance is an important factor in determining the operating point of the fan. The system resistance is generally calculated by an equivalent circuit program or a numerical analysis based on the empirical formula. The air flow rate of the fan operating point of a 5 MW class high voltage generator is about 5.94 m^3/s and the static pressure is about 1087 Pa. We provided the system resistance curve according to the fan operating point to a fan specialist company and designed four kinds of fans according to the shape of the blades. We performed a numerical analysis and an ANSI/AMCA standard 210-99 specification fan performance test to verify the performance of the fans. We used the ANSIS ICEM-CFD grid generation program to analyze the fan performance characteristics through a numerical analysis, and generated about 6 million grids. We expected turbulent flow so we used the STANDARD model. The comparison of the performance curves through the numerical analysis and the performance tests. As a result of the comparison, we could obtain a reliable analysis result.

RAPID PROTOTYPING OF THE SMALL SCALE CSP FAN

David VOLPONI Sapienza University of Rome (Italy)
Michael, B. WILKINSON Stellenbosch University (South Africa)
Giovanni DELIBRA Sapienza university of Rome (Italy)
Alessandro CORSINI Sapienza University of Rome (Italy)
Sybrand, J. VAN DER SPUY Stellenbosch University (South Africa)
Theodor W. VON BACKSTRÖM Stellenbosch University (South Africa)

The MinWaterCSP project aims to reduce the water consumption of concentrating solar power (CSP) plants by 75 to 95 % relative to wet cooling systems and to improve plant efficiency by 2 to 3 % relative to current dry-cooled systems by introducing novel dry/wet cooling technology. This hybrid cooling system will make use of large axial flow fans to condense the process fluid.

As part of the project a new high efficiency, reduced noise axial flow fan was developed and manufactured for a 24 ft. full scale test facility at Stellenbosch University (constructed as part of the MinWaterCSP project). This fan is referred to as the M-fan. The M-fan was designed with an optimised hub-tip ratio and minimised exist kinetic energy velocity profile. At its design flowrate of 333 m^3/s and a blade setting angle of 34°, full scale 3-D CFD simulations predicted a pressure rise of 116.7 Pa at a fan static efficiency of just below 60 %.

A 1.5 m diameter scaled model of the M-fan was manufactured and tested in the BS848 fan test facility, also at Stellenbosch University. The test results showed a fan static efficiency at the design point of 59 %, for a tip clearance of 2 mm, at a blade setting angle of 35°. Measurements performed on the 1.5 m fan do however indicate a small discrepancy in its effective, as-manufactured designed blade angle.

To confirm the design process for the M-fan, its outlet velocity profile was measured on a 630 mm diameter ducted fan test facility. The measurements included measuring the fan performance characteristics, as well as the inlet and outlet velocity profiles. The 630 mm diameter fan blades were 3D printed in ABS into two parts, glued together, and in the end mounted to an aluminium hub. In this case, this procedure guaranteed the maximum prototype cost reduction. The results showed that the small scale fan can replicate the operating point of the 24 ft fan albeit at a slightly lower efficiency due to the effects of a different installation. All fan tests were fans performed at a tip speed of 58 m/s.

A comparison of the 1.5 m and 630 mm diameter fan test results to performance values scaled from the 24 ft. CFD simulation data, indicate that accurate correlation of data is dependent on the accuracy of the fan manufacturing process, the accuracy of measurements performed and the CFD process used for the simulation of the fan.

STUDY OF SOUND SOURCE DISTRIBUTION IN CENTRIFUGAL FAN USING TRANSPARENT WALL

War War MIN SWE	Mandalay Technological University (Myanmar (Burma))
Tetsuya OKUMURA	Nagasaki University (Japan)
Atsuhiro KOYAMA	Nagasaki University (Japan)
Hidechito HAYASHI	Nagasaki University (Japan)
Makoto HIRATA	Nagasaki University (Japan)
Wakawa NOGAMI	Panasonic Eco Systems (Japan)

Sound source investigation is very important for reducing the fan noise. The sound photograph and beam forming method are recently used for this purpose. The axial fan is able to use this method because the impeller is directly observed from the microphone array. But at the case of centrifugal fan, the impeller is hidden in the casing or shroud and it is difficult to use those method directly. For example, it is used for the case at free from casing, but it is restricted only the open operation not to pressurized condition.

This research is focused to make clear the sound source distribution in the centrifugal fan with pressurized operation. For this purpose, It is used the transparent wall. The transparent wall is rolled the wall for the flow, but not wall for the sound. Transparent wall is researched by some researchers for the reduction of the noise and know the simple sound source map. But the application of the complex case for the fan is not known. In this research, it is cleared the characteristics of the transparent wall for the pressurized conditions. It is cleared that the transparent characteristics is varied so much at the pressurized condition.

Next the application for the pressurized condition of the tubular centrifugal fan is researched. The fan is consisted of the centrifugal impeller and axisymmetric casing. The air flows onto the impeller from the center of impeller and turns to radial direction in the impeller. The radial directional flow makes the large pressure rise in the impeller. After the impeller, the air turns to axial direction again. The flow pattern in the tubular fan is so complex that the large loss and noise are generated in the fan. We analyze the flow characteristics of the tubular fan with numerical simulation.It is attempted the sound transparent wall for the casing to know the sound source. The low frequency sound of about 500 Hz that is estimated the source being the impeller is clearly emerged from the outlet of the impeller and the high frequency noise that is estimated the source location of the inlet leakage flow is mainly radiated from the inlet of the fan.

COMPARISON OF MULTIPLE FAN SYSTEM ASSEMBLIES USING AN ACOUSTICALLY TRANSPARENT DUCT

Kristóf TOKAJI Budapest University of Technology and Economics (Hungary)
Balázs SZEKER Budapest University of Technology and Economics (Hungary)
Csaba HORVÁTH Budapest University of Technology (Hungary)

Fans are widely used for industrial as well as household applications. In many of these cases, for instance air conditioning or ventilation systems, the fans are built into ducted environments. The number of such applications surrounding us in our daily lives has increased with the advancement of technology, and therefore an eye needs to be kept on noise pollution levels, which need to be reduced further in order to meet the requirements of strict laws and the demands of consumers. In order to do so, the noise generation mechanisms of fans have to be investigated, understood, and inevitably eliminated. Combining an array of microphones with beamforming technology, one can localize noise sources and determine noise generation mechanisms.

In the case of ducted fans, such as the one investigated herein, acoustic investigations should be carried out in a ducted environment in order to guarantee proper flow conditions. One approach could be to mount microphones on the wall of the duct. Such measurements run with many complications, including the disturbing effects of duct modes and flow induced microphone self-noise. Another approach would be to place the array of microphones outside of the flow (outside of the duct) and to use an Acoustically Transparent Duct (ATD) in order to gain direct access to the noise radiating from the source (eliminating the effect of duct modes). Such an ATD was designed by our research team, and has been shown to allow through acoustic signal which are sufficient for investigating low speed fans with beamforming technology for a wide frequency range while providing a hydrodynamically impenetrable duct surface for the flow. In our design, a perforated duct section provides the ducts shape and connects to other components (for example the fan), while a layer of stretch film, characterized by low acoustic impedance, provides hydrodynamic impenetrability. This article is one in a series regarding the ATD designed by our research team. While the first articles investigated the acoustic characteristics of the ATD and its applicability [1, 2], this is the first test case, where the focus of the investigation is on studying fan noise sources.

In this investigation the most commonly occurring configurations, which can be built from a fan, the ATD, an inlet cone and spiral wound duct sections (ex. free-inlet free-outlet, ducted-inlet ducted-outlet, etc.) are looked at. The various components of the configurations have differing effects on the flow, and therefore also on the acoustic characteristics. The examination and comparison of the various cases helps to define the effect of each component on the noise generation mechanisms of the fan. In our research we examined the effects of each component on the classical fan noise generation mechanisms found in the literature, such as turbulence ingestion noise, boundary layer noise, trailing edge noise, vortex shedding noise, separation noise and tip-leakage flow noise.

COMPARISON OF MICROPHONE ARRAY METHODS FOR THE CHARACTERIZATION OF ROTATING BROADBAND NOISE SOURCES

Christof OCKER	Aalen University (Germany)
Gert HEROLD	Technische Universität Berlin (Germany)
Florian KRÖMER	Friedrich-Alexander University Erlangen-Nürnberg (Germany)
Wolfram PANNERT	Aalen University (Germany)
Ennes SARRADJ	Technische Universität Berlin (Germany)
Stefan BECKER	Friedrich-Alexander University Erlangen-Nürnberg (Germany)

The localization of rotating sound sources with a microphone array is a major task to reduce broadband axial fan noise. Depending on the adjusted operating point and considered frequency band, the sound sources are located on the leading or trailing edges of the fan blades. Considerable effort has been put into compensating the relative motion of the sound sources to the microphone array and to understand the rotating sound source mechanism. Motion compensation can be done in time domain or frequency domain. In the time domain, the emitted signals from the moving sources are reconstructed by continuously moving the focus of the microphone array along the rotational motion, taking into account the time delays and the Doppler effect due to the rotation. In the frequency domain, the signals are first Fourier transformed. Afterwards, the resulting cross spectral matrix is multiplied with the steering vectors to shift the phase of the microphone signals according to the focus points. Motion compensation in the frequency domain can be done in different ways, e.g. employing virtual rotating microphones to transform the pressure signals into a rotating reference frame or modal decomposition of the rotating sound field.

After motion compensation, beamforming is used to localize the sound sources. For some standard and high-resolution beamforming methods, the cross spectral matrix is needed. The cross spectral matrix can be calculated in the time domain with Welch's method or in the frequency domain with Daniell's method.

This paper contains a comparison of the results of different methods for motion compensation. At first, a simulated benchmark case with discrete rotating sound sources is considered. The evaluation is done with respect to the correct estimation of the source position and source amplitude. Based on these findings, recommendations of the number of microphones to be used for actual measurements on rotating systems are given. In a second case, a benchmark fan is analyzed for different frequency bands to compare the source distribution and source amplitudes. Advantages, disadvantages and limits of the considered methods are shown.

CFD ANALYSIS OF AXIAL FLOW FANS FOR CSP AIR-COOLED CONDENSERS

David VOLPONI Sapienza University of Rome (Italy)
Michael WILKINSON Stellenbosch University (South Africa)
Johan VAN DER SPUY Stellenbosch University (South Africa)
Tomasso BONANNI Sapienza Univerisity of Rome (Italy)
Lorenzo TIEGHI Sapienza University of Rome (Italy)
Giovanni DELIBRA Sapienza University of Rome (Italy)
Alessandro CORSINI Sapienza University of Rome (Italy)
Theodor W. VON BACKSTRÖM Stellenbosch University (South Africa)

The MinWaterCSP project aims to reduce the water consumption of concentrating solar power (CSP) plants by 75 to 95 % relative to wet cooling systems and to improve plant efficiency by 2 to 3 % relative to current dry-cooled systems by introducing novel dry/wet cooling technology. This hybrid cooling system will make use of large axial flow fans to condense the process fluid.

As part of the project a new high efficiency, reduced noise axial flow fan was developed and manufactured for a 24 ft. full scale test facility at Stellenbosch University (constructed as part of the MinWaterCSP project). This fan is referred to as the M-fan. The M-fan was designed with an optimised hub-tip ratio and minimised exit kinetic energy velocity profile. At its design flowrate of 333 m^3/s and a blade setting angle of 34°, full scale 3-D CFD simulations using ANSYS Fluent predicted a pressure rise of 114.7 Pa at a fan static efficiency of just below 60 %. This full-scale simulation of the fan was also performed using the actuator disk method and indicated a fan static pressure rise of 111.4 Pa and a fan static efficiency of 63.1 %.

The M-fan was also simulated at a diameter of 1.5 m diameter. At a design flowrate of 14.4 m^3/s and a blade angle of 34° it indicated a fan static pressure rise of 116.5 Pa , and fan static efficiency of 57.9 % . The fan is also simulated with a range of tip gaps which are found to reduce fan performance.

Concurrent RANS computations were also carried out using the simpleFoam solver of the OpenFOAM library for CFD computations with the non-linear closure of Lien et al. to further verify the performance of the fan and analyse the flow field in the blade-to-blade passage.

Two other fan configurations were also designed, namely the L3C2-fan and the T-fan. They where were designed for reduced noise signature and improved efficiency by adjusting the exit velocity profile, compared to the M-Fan. In particular, the L3C2 was obtained by optimizing the chord and pitch distributions of the M-Fan for reduced trailing edge noise, leading to a completely different shape of the rotor blade. With the T-Fan the design paradigm of the rotor was changed and a forced-vortex distribution of the work along the blade span was enforced to reduce the load at the tip of the blade. CFD computations were carried out also on these configurations, to verify performance of these new rotors.

AEROACOUSTIC AND AERODYNAMIC PERFORMANCE OF SILENCER UNITS AT DIFFERENT INFLOW PROFILES

Johannes WALTER Karlsruhe Institute of Technology (Germany)
Martin GABI Karlsruhe Institute of Technology (Germany)

Silencer units are installed downstream of axial fans to reduce the noise emission. An inhomogeneous or swirling fan outflow profile, respectively inflow profile to the silencer unit, leads to increased aerodynamic losses. Additionally, the acoustic properties are impaired by the sound generation of areas with high flow velocity.

Experiments are carried out on a scaled test rig, which is based on a state of the art configuration. Total and static pressure measurements are conducted in the diffuser, the silencer and downstream the silencer. The results show that there is a strong dependence of the performance of the configuration on the diffuser inlet profile. The undisturbed or hubstrong inflow profile leads to the formation of a high velocity area in the center of the splitter silencer. The tipstrong inlet and swirl inflow profiles lead to the opposite behavior. There are high velocities close to the casing and low or negative velocities in the center. The loss coefficient of the silencer unit can increase up to 169 % depending on the inflow profile. The flow noise, which is generated at the outlet of the splitters, is investigated as well. Therefore two characteristic silencer flow profiles are reproduced. Additionally, a nearly homogeneous flow distribution in the silencer is investigated as a reference. The results show that the flow noise for the characteristic silencer flow profiles is up to 11 dB higher than for the reference flow profile.

OPENFOAM APPLICATION ON PERFORMANCE PREDICTIONS OF AN INDUSTRIAL CENTRIFUGAL FAN WITH AIRFOIL BLADES

Jing WANG Daltec Process Fans (Canada)
Marcel KAMUTZKI Daltec Process Fans (Canada)

Computational fluid dynamics (CFD) has increasingly been applied to industrial fan design over the past decade. The high-performance computation for CFD has become more accessible as the cost of computer hardware has been cheaper. The well-known finite volume CFD solver, OpenFOAM widely used in schools is gradually being a favourite approach by industrial fan designers for predicting the performance of industrial fans for it is fully open-sourced and no limitations are on the parallel computation. This paper presents a modelling approach of an airfoil bladed centrifugal fan which we developed for performance predictions of our regular industrial products. In the model, the snappyhexmesh tool is applied to create the hexahedral dominant mesh with boundary layers on the fan wall surfaces. The multiple reference frame (MRF) model is employed for the simulation of rotation zone. The Spalart-Allmaras model and k-ω Shear Stress Transport (SST) model are used for the turbulence flow for a comparison purpose. The OpenFOAM allrun template for fan performance predictions are developed. The parallel computation setup and postprocessing programing are also presented in the paper. The results of fan pressure, power, and efficiency using the two turbulence models are compared and discussed and the future development on the CFD model is recommended.

Keywords: Industrial centrifugal fan, Fan performance prediction, Open source solver, CFD

SEPARATION OF TONAL AND BROADBAND NOISE COMPONENTS BY CYCLOSTATIONARY ANALYSIS OF THE MODAL SOUND FIELD IN A LOW-SPEED FAN TEST RIG

Maximilian BEHN German Aerospace Center (DLR) (Germany)
Benjamin PARDOWITZ German Aerospace Center (DLR) (Germany)
Ulf TAPKEN German Aerospace Center (DLR) (Germany)

The development of axial turbomachinery such as ventilators and aero engines imposes increasing requirements regarding a reduction of the tonal and broadband sound field components. The analysis of tonal sound field components can be performed by use of the well-established Radial Mode Analysis, which allows the determination of the sound power of all cut-on modes on the basis of sound pressure measurements using in-duct microphone arrays. Recently, the application of the Radial Mode Analysis to determine broadband mode amplitudes for experimental data from a laboratory low-speed fan test rig was presented. The method assumes that the broadband sound field can be modelled as a quasi-stationary process. In this study, another approach will be investigated which is based on the assumption, that the statistical properties of the broadband sound field are periodic in time with respect to the rotor revolution, i.e. the assumption of a cyclostationary process.

In contrast to the quasi-stationary analysis, the cyclostationary analysis allows the separation between tonal and broadband sound field components without affecting the amplitude and phase relations of the respective signal components. The effect of this approach on resulting mode spectra, sound powers and mode coherences will be discussed using experimental data measured at a laboratory low-speed fan test rig. Furthermore, the potential benefits of the cyclostationary analysis for the development of a source localization technique for rotating sources will be examined. Based on the cyclostationary analysis the Wigner-Ville spectrum can be computed, which describes the development of the statistical signal properties such as auto- and cross-power spectra over the rotor revolution. As a result, the positions and strengths of rotating sources can be tracked. The identification of the modulation of sources and the separation of rotating and static sources in the case of a rotor-stator stage are two main advantages that result from the cyclostationary analysis.

ENERGY DISSIPATION IN THE AXIAL FAN TIP CLEARANCE FLOW

Matej MILAVEC Hidria Rotomatika d.o.o. (Slovenia)
Stanislav PIVK Hidria Rotomatika d.o.o. (Slovenia)
Daniel VIDAL DE VENTOS Hidria Rotomatika d.o.o. (Slovenia)
Branko ŠIROK Faculty of Mechanical Engineering Ljubljana (Slovenia)
Benjamin BIZJAN Abelium d.o.o. (Slovenia)

A study of velocity and pressure fluctuations in the tip clearance flow of an axial fan for two different rotor blade tip designs is presented. Results determined with the standard blade tip design are compared to those from modified blade design with a tip winglet. Comparison of integral sound parameters indicates significant noise level reduction for the modified blade tip design. Noise reduction depends on the topology of flow structures in the area of gap-flow with the emphasis on the decay of vortex structures from larger to smaller local – time scales. In order to observe this phenomenon, a new experimental method was developed. This method is based on simultaneous measured values of local flow velocity and pressure. It enhances the identification and determination of the local flow structure and its turbulent flow properties. Besides, energy distribution within the phase space together with generation and dissipation of the turbulent kinetic energy within the gap-flow determines characteristic sources of acoustic emission of the fan. Locations of attractors within the phase space were also determined in order to support basic assumptions and conclusions:

Transition from integral operating points of the fan with minimum to those with maximum air-flows significantly affects the topologic structures of attractors; it is quite deterministic and linked with the expected conversion of the pressure into kinetic energy during the period that coincides with the position of the blade tip passage over the observed sensor. The following inter-blade areas exert larger velocity fluctuations typical for decay of larger eddy structures and are consequently closely linked with wideband noise generation. Emitted sound power depended significantly on the shape of tip presented here. The largest difference coincides with the best operating range of the fan where best efficiency and the lowest noise level were observed. The relation between turbulent kinetic energy production and emitted noise level was explained on a phenomenological level.

A CYCLOSTATIONARY APPROACH IN EXTRACTING MODULATION FEATURE FROM FAN VIBRATIONS

Ning CHU Zhejiang University (China)
Shyang LI Zhejiang University (China)
Dazhuan WU Zheijang University (China)

Vibration signals related to fan aerodynamic performance are generally transient, non-stationary and non-linear. To extract aerodynamic-related features from these complicated signals, this paper presents an improved cyclostationary approach and employs an enhanced envelop spectrum to detect the modulating, harmonic and coupled frequencies, as well as to reconstruct principle modulating intensities. These modulation-related components have strong links with the flow-induced excitation, and can provide an alternative way to reveal the fan aerodynamics. Finally proposed approach is validated by typical simulation and axial-flow fan experiments.

EFFECTIVENESS OF BLADE FORWARD SWEEP IN A SMALL INDUSTRIAL TUBE-AXIAL FAN

Massimo MASI University of Padova (Italy)
Andrea LAZZARETTO University of Padova (Italy)
Stefano CASTEGNARO University of Padova (Italy)

It is well-known that forward sweep in the radial stacking line of rotor blades for low-speed axial fans generates an appreciable gain in the stall margin at the expense of a reduction of the fan pressure at design point. In addition, several rotors with increasing blade circulation from hub to tip show a small percentage gain in the maximum fan efficiency. Some methods were suggested in the literature to include the sweep angle already in the preliminary design, in order to limit the extensive Experimental and/or Computational Fluid Dynamics trial and error work required to find the distribution of sweep angle best suited to a specific fan design. The authors presented few years ago a technique to increase either the performance or the efficiency of an existing arbitrary vortex design tube-axial fan by introduction of a uniform radial distribution of blade forward sweep. According to the CFD calculations performed for a 560 mm tube-axial fan, this technique was able to increase the fan pressure at design point without decreasing the efficiency. The superimposition of 6 degrees of forward sweep at the blade tip, in addition to the distribution suggested by this technique, resulted in a significant increase of the maximum fan pressure.

The paper presents the results of experimental tests performed on a 315 mm tube-axial fan similar to the fan considered in the CFD analyses. The fan features a hub-to- tip ratio equal to 0.4 and a tip clearance typical for industrial fans of this size. To quantify the effects of blade forward sweep on the performance and efficiency figures of this small tube-axial fan, three different stacking line of the rotor blades have been tested: unswept, forward swept and forward swept with additional sweep at the blade tip. Experimental data prove the effectiveness of the design method for these small fans.

COOLING OF ELECTRICAL MOTORS

Georg KLEPP HS-OWL (Germany)
Alexander PRIES HS-OWL (Germany)

The efficiency of electrical drives and motors can be enhanced by an efficient cooling. In this context the cooling of a three phase asynchronous motor, 2 pole, 1.3 kW, IE2, size 80 is exemplarily investigated. The cooling of the motor is by a radial impeller with straight blades on a plate. There are different sizes, shapes and numbers of the blades. There are different sizes and shapes for the plate. The (basically) cylindrical fan cover guides the airflow exiting the impeller in axial direction in order to cool the motor. Different geometries for radial impeller and fan cover were investigated in order to improve the motor cooling.

In experimental setups, the volume flow rate generated by the impeller as well as the motor temperature due to different efficiencies in cooling were investigated. Different commercial available impellers as well as own designs (produced by rapid prototyping (i.e. 3D printing)) were tested.

In numerical simulations (three dimensional transient incompressible Reynolds averaged Navier-Stokes (RANS) equations) the flow field for different impeller geometries was computed, looking for an optimal design for the motor cooling. In addition the resulting flow field was analysed to get an insight into the evolving flow patterns.

If only the impeller geometry is analysed (the fan casing and end cap geometry are fixed), the most influential parameter is the impeller diameter: with increasing diameter the fan performance increases but also the gap between impeller and fan cover decreases thus decreasing the volume flow rate and cooling efficiency. A blade number of eleven seems optimal. The influence of the other geometric parameters (shape and size of blade and plate) is of secondary importance.

The analysis of the computed flow fields reveals the critical design parameters for the motor cooling: the flow is choked in the gap between impeller and fan cover and the gap between fan cover and motor casing (end cap). An optimisation of the motor cooling has to involve the redesign of the fan cover and motor casing in order to achieve a significant increase in cooling efficiency.

EXPERIMENTAL INVESTIGATIONS OF MIXED FLOW FANS WITH VARIOUS DOWNSTREAM ANGLES

Anne TREDER Technical University of Berlin (Germany)
Paul Uwe THAMSEN Technical University of Berlin (Germany)

Axial and centrifugal fans are represented in a wide variety of different shapes on the market. If the design point requires large flow rates with a sufficiently large increase in pressure, the recommended design point is within the range between the axial and radial fan type, in the area of mixed flow fans. Mixed flow fans are still a rarity. Only a very few recommendations can be found in the literature for the design of mixed flow fans and therefore also no empirical coefficients for the diagonal design are available.

A special field of application for mixed flow fans is the cooling of motors and electrical components. Motor cooling fans are installed directly on the motor shaft inside the housing and ensure the removal of the heat, yielded by engine components, into the passing air flow. Major demands for the design point of these fans are large flow rates with the associated high flow velocities for sufficient cooling capacity. The airflow through the engines, especially through the narrow gap between the stator and rotor, causes high pressure losses. Thus, the requirements for the cooling fans are high flow rates with comparatively high pressure increases.

Similar to centrifugal fans, there is also a flow deflection from axial to radial direction in mixed flow fans. The risk of flow separation at the shroud is increased because of the relatively wide blade channels. In order to improve the flow deflection and to prevent the shroud separation, different shroud contours with an increased radius of $R130$ and $R180$ were investigated.

The investigations revealed that a too sharp flow deflection with wide blade channels, such as in R80, cause flow separation at the shroud, while a smooth radius of R180 results in increased friction losses and deteriorated efficiencies. Furthermore, the impact of various downstream angles chi = 30° (almost axially) to chi = 90° (centrifugal) was evaluated for the volume flow and the pressure increase. Changing the downstream angle to smaller chi decreased the flow rates with smaller pressure rises.

Compared to axial fans, the mixed flow fan type already achieves higher pressure rises in a single stage and while operating without a stalling dip. Mixed flow fans can be easily integrated in fluid systems with axially parallel inlet and outlet flow, as well as axial fans and therefore represent a serious alternative in the application field between axial and radial fans.

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF AXIAL FAN AEROACOUSTICS AT DISTURBED INFLOW CONDITIONS

Andreas LUCIUS	ebm-papst Mulfingen GmbH & Co. KG (Germany)
Simon HÜTTER	ebm-papst Mulfingen GmbH & Co. KG (Germany)
Philipp DIETRICH	ebm-papst Mulfingen GmbH & Co. KG (Germany)
Marius LEHMANN	ebm-papst Mulfingen GmbH & Co. KG (Germany)
Marc SCHNEIDER	ebm-papst Mulfingen GmbH & Co. KG (Germany)
Thomas GEYER	BTU Cottbus (Germany)

In customer applications axial fans are usually operating at disturbed inflow conditions. A typical inflow disturbance is a heat exchanger upstream the fan, which is mounted in a box. A simplified configuration of this installation is the fan with a rectangular box in front. This test case is investigated experimentally at ebm-papst using standardized aero acoustic measurements, microphone array and flow visualization of a 5 blade axial fan with 910 mm diameter. The inflow disturbance increases the broadband noise level over the whole spectrum. Especially in the low frequency range up to 1000 Hz the sound power increases around 10 dB. There is no noticeable effect on fan performance. Pressure rise as well as efficiency is nearly the same with and without box in front.

The effect of increased noise is investigated via CAA simulations of the axial fan with and without box. For this purpose LES simulations are done using FW-H propagation for far field aero acoustics. As emission surfaces the rotating blades as well as a permeable emission surface on the fan suction are applied. Simulations accurately predict the difference in broadband noise up to 4 kHz. The absolute noise levels are also well predicted, with a slight over prediction of 5 dB in the range of 2000 to 4000 Hz. Due to short simulation time of 2 rotor revolutions, the effect on tonal noise is not resolved in the simulation.

The simulations indicate that large scale vortical structures separate from the corners of the box. These structures unite to large regions of high turbulence at roughly middle of top and bottom walls of the box. This area of increased turbulence interacts with the fan blades. The existence of these large scale structures is supported experimentally by sound source localization with array technique and flow visualization on the box surface. Measurements as well as CAA show increasing sound sources at the leading edge every time the blade passes the vortex.

HUMMING NOISE MECHANISM IN AUTOMOTIVE BLOWER

Manuel HENNER	Valeo Thermal Systems (France)
Aurélien SERAN	Valeo Thermal Systems (France)
Bruno DEMORY	Valeo Thermal Systems (France)
Saïd NAJI	Valeo Thermal Systems (France)
Olivier CHÉRIAUX	Valeo Thermal Systems (France)
Fabrice AILLOUD	Valeo Thermal Systems (France)
Thibaud MATHARAN	Valeo Thermal System (France)

Automotive blowers for HVACs units are highly constrained ventilation devices, mainly for the purpose of the integration in the dashboard which require a good compactness. As a consequence, many efforts have been put in the reduction of both the wheel diameter and the scroll width, and the machines in production represent now a good compromise between aerodynamic efficiency, acoustics and packaging.

However, some new trends related to the cabin comfort are bringing an additional pressure loading on the blower, which has to force the air through a series of heat exchangers (heating and cooling), air filters and flaps aimed to redirect the flow. In some circumstance, a spurious additional humming noise is observed in the system and it has been named by product engineers as a “vuvuzela” noise. This particular phenomenon is quite noisy and unpleasant and can be easily identified in the acoustic spectrum with the evidence of wide peaks at the wheel rotational frequency and its harmonics.

Some acoustic tests have been conducted on a test rig at different flow rates and for two different geometries. It shows that the humming noise appears when the wheel is highly loaded, and when interactions between the wheel and the scroll yield to a large amount of recirculation.

Several unsteady simulations have been performed for the same conditions and have allowed a deeper analysis of the flow. They highlight the effect of the recirculation at the top of the wheel, which brings some vortical structures at the wheel entrance and which impacts the blade leading edges. Simulations show the appearance of these vortexes which are spinning at low velocity inside the wheel, and which are creating quasi periodic pressure fluctuations on wheel and scroll surfaces. Additional pressure recordings at several locations confirm the relationship between the wide peaks and the vortex distributions.

These analyses have allowed to identify the phenomena and to bring some technical solutions. This method will be used also to drive the design of the next generation of blowers in order to remove the occurrence of this humming noise mechanism.

EXPERIMENTAL INVESTIGATION OF INSTALLATION EFFECTS IN A LOW-SPEED FAN SYSTEM

Joachim DOMINIQUE von Karman Institute (Belgium)
Julien CHRISTOPHE von Karman Institute (Belgium)
Christophe SCHRAM von Karman Institute (Belgium)

The reduction of the noise generated by industrial fans that is imposed by ever increasing regulations requires an improved understanding of aeroacoustic mechanisms, in particular of the interactions taking place between the components of HVAC systems or household appliances. Indeed, while the noise produced by an isolated blower is usually characterized under ideal inflow conditions with uniform inlet profile and low turbulence intensity, such conditions are in practice rarely met in the full system, due to product compactness and integration constraints. This leads to extraneous noise generation that can compromise the acoustic performance of the equipment, and thereby increase its development time and cost.

The paper describes an experimental investigation of installation effects for a low-speed fan, based on an advanced multi-ports modal eduction technique. This approach, implementing arrays of microphones and loudspeakers, permits discriminating between aerodynamic and acoustic effects and hence to isolate the influence of flow distortions located upstream of the fan. The outcome of the procedure is a quantitative measure of the noise that is produced by the fan, irrespectively of artifacts of the test rig that would inevitably contaminate the results otherwise.

The cornerstone of the procedure is the projection of the acoustic field measured by the microphone arrays on a modal basis corresponding to the Green's function of the circular duct, parameterized by its axial wavenumber, radial and azimuthal mode orders. In this study, the planar and first two azimuthal modes are included in the analysis. The modal acoustic reflection of the test rig terminations, as well as the modal scattering properties of the fan, are obtained using the loudspeaker arrays.

The results provide a quantitative assessment of the types of distortions that cause the most significant noise increase depending on their geometry and expected inflow distortions

MEASURING THE FLOWRATE OF FANS BY EMBEDDED SENSORS

Alain GUÉDEL CETIAT (France)
Isabelle CARÉ CETIAT (France)

In most ventilation applications a measurement of the flowrate allows a better control of the system (especially when the flow varies with time) and a possible reduction of the energy consumption by avoiding to oversize the fan. However, the actual flow performance of a fan in a system depends on the overall pressure loss of the system, which is often not accurately estimated, and on potential fan system effects. The use of the technical specifications given by the fan manufacturer (e.g. the fan curve) may be not sufficient in this case to reach the expected accuracy on the flow rate knowledge.

A way to measure the flowrate of the system may be to do it on the fan itself. Fans with embedded sensors can be a suitable solution for this flow measurement.

In this paper different solutions based on the measurement of the differential pressure on the bell mouth of a double Inlet forward curved centrifugal fan of 200 mm diameter have been investigated. The tests were performed at different speeds and fan operating points. They included measurements with 9 pressure taps on each inlet bell, then with a pressure probe patented by the fan manufacturer Nicotra Gebhardt. This probe was fixed on one of the fan inlet arms. The choice of the arm and the radial location of the probe have been investigated to estimate their influence on the flow measurement.

All these tests have been performed on a test rig using orifice plates which allows to know the actual flow rate with a good accuracy. A relationship between the known flow rate and the measured differential pressure has been established for the two configurations:

• pressure measurements on the bell mouth taking into account the number of pressure taps,

• pressure sensor on an inlet arm taking into account the radial location of the probe and the circumferential position of the arm itself.

Considering the measurement uncertainty of the instruments used the uncertainty on the established relationship has been estimated for each test configuration.

A simulation has then been performed to estimate the uncertainty on the flow rate measurement in case of on-site use taking into account the possible accuracy of the pressure sensor, the knowledge of the actual air density and the impact of some fan installation effects.

DESIGN AND TESTING OF AN ISO 5801 INLET CHAMBER TEST RIG AND RELATED ISSUES WITH THE STANDARD

Stefano CASTEGNARO University of Padova (Italy)
Massimo MASI University of Padova - DTG (Italy)
Andrea LAZZARETTO University of Padova - DII (Italy)

A rig has been designed and constructed according to the ISO 5801 standard, to test fans with a maximum inlet diameter equal to 0.8 m. The rig features an inlet chamber configuration and a multi-nozzle system for the flow-rate measurements. Before starting the design process a deep study of the ISO Standard has been carried out and two issues emerged: i) even if the requirements of all the components and rig parts are identified, a direct design procedure is not available in the Standard or in the literature; ii) some incoherent informations exist within the Standard. This last aspect is particularly tedious for the rig designer, because these ambiguities translate into a deeper analysis of the flow phenomena and a consequent interpretation of the Standard. These issues increase the time required for the design process and, in turn, the costs. Regarding the positioning of the nozzles on the wall between the two chambers for instance, the Standard does not clarify if a symmetrical allocation (with respect to the chamber longitudinal axis) is mandatory or recommended. Another unclear aspect deals with the operational differences among inlet chambers of type 1, 2, and 3.

In this work the design and construction of the ISO 5801 test rig installed at the University of Padova, Italy is described. The rig has been designed for a maxium flow-rate of ~ 8 m3/s and for a maximum negative pressure of 10000 Pa (~0.1 atm). The rig design has been organized into a direct procedure that can be easily applied by anyone interested in designing a rig with this common configuration. Furthermore, all the issues related to the Standard and encountered during the design process are highlighted and discussed.

The commissioning of the rig is presented as well; leakage tests were performed with the two-phase technique suggested by the Standard. Moreover, a case-study 800 mm Vane-Axial fan has been tested with different fan and rig configurations; the data collected are presented and discussed.

The aim of the paper is twofold: i) providing to anyone who is interested in designing an inlet chamber test rig with a direct procedure to perform satisfactorily the design of the facility, and ii) focusing the attention on some issues contained within the ISO 5801 Standard and suggesting possibile corrections/improvements.

Designing, Building and Validating a Test Rig to Measure Two-port Data of Small Fans

Emil GARNELL KTH Royal Institute of Technology (Sweden)
Mats ÅBOM KTH Royal Institute of Technology (Sweden)
Guy BANWELL Dyson (United Kingdom)

There is a great need in the industry for measurement methods to characterize acoustic sources in ducts. One way to obtain a complete description of a source is to measure 2-port data, comprising of the source scattering matrix and the source vector. The resulting model can then be used to predict the source properties, e.g., radiated sound power, in the plane wave range for all installation conditions. Methods to measure the two-port model have been developed over the last few decades and can today be efficiently used for industrial purposes. The present paper offers a review of the main guidelines for designing and building a two-port rig to measure high speed small fans, as well as an example of how to use the data to predict the noise emission of a product.

All rig elements have been designed after a literature review and an analysis of the physical principles governing the behavior of the rig. Guidelines on microphone spacing, loudspeaker mounting, rig terminations and overall rig dimensions are given.

The theory behind the measurement method of the active two-port in a duct is presented. Additionally, a number of different post-process methods are evaluated with respect to the properties of the experimental setup used i.e. the number of available microphones, the magnitude of the reflection coefficient at the rig terminations and the type of test object measured. The standard method that is most widely used nowadays is shown to become singular when the reflection coefficients at the rig terminations are high. A new post-processing method is suggested, and tested against the standard one. It is shown to behave better in highly reflective cases.

NEW METHODOLOGIES FOR EFFECTIVE MARKET SURVEILLANCE OF LARGE FANS WITH THE INTAS PROJECT

Christian Holm CHRISTIANSEN Danish Technological Institute (Denmark)
Ingrid WEISS WIP Renewable Energies (Germany)
Tomas JEZDINSKY European Copper Institute (Belgium)
Paul WAIDE Waide Strategic Efficiency (United Kingdom)
Nerea RUIZ FUENTE European Environmental Citizens' Organisation for Standardisation (Belgium)

The goal of the INTAS project, funded by the EU's Horizon 2020 programme, is to provide technical and cooperative support, as well as capacity building activities, to Market Surveillance Authorities (MSAs). The need for the INTAS project arises from the difficulty that MSAs and market actors face in establishing and verifying compliance of large industrial products. Large fans are subject to energy performance requirements under the Ecodesign Directive and are one of the two products in the project's focus.

Specifically, INTAS aims to:

- Support European Member State MSAs deliver compliance for large fans;
- Support industry to be sure of what their obligations are under the Ecodesign Directive and to deliver compliance that will be broadly accepted by MSAs;
- Foster a common European approach to the delivery and verification of compliance for large fans.

In a first stage, INTAS has analysed the existing testing avenues in Europe and beyond, and explored test standards, facilities, procedures and methods already in place for large fans. Now, it is in the phase of defining an effective compliance framework for MSAs and manufacturers and will also conduct real evaluation and testing exercises.

Testing and evaluation methods INTAS is considering at present include:

1) Those based on current industry practices for which INTAS will investigate the strengths and limitations of:

    a/ scale-model testing, i.e. testing a smaller kW fan of the same design as part of series of larger fans and extrapolate findings for the larger models of the same design

    b/ part load testing, i.e. testing a fan in a part load operational point and calculating/extrapolating performance at the best efficiency point and seeing if the performance of such approaches in independent laboratories might be suitable for Ecodesign verification purposes.

2) Participation in witness tests or factory acceptance tests to define which procedures could be specified for MSAs to assist at manufacturers' premises tests.

3) Evaluation of a consolidated approach for auditing manufacturers conformity assessment procedures (Art. 8 of DIR. 2009/125/EC) to define recommendations on required documentation of conformity assessment, in alignment with MSAs and industry.

INTAS will finally evaluate the results of the previous processes and ensure that the proposed methodology is valid and reliable for national MSAs.

Throughout the entire project, INTAS will foster market surveillance collaboration between MSAs and raise awareness and information exchange of the product energy performance and market surveillance among key stakeholders.

A Statistical Survey on the Actual State-of-the-Art Performance of Radial Flow Fans Based on Market Data

Nicola ALDI University of Ferrara (Italy)
Nicola CASARI University of Ferrara (Italy)
Michele PINELLI University of Ferrara (Italy)
Alessio SUMAN University of Ferrara (Italy)

In the current energy and environmental context, sustainability is emerging as the new paradigm of reference. A recent European Directive establishes a framework for the setting of eco-design requirements for energy-related products, in order to reduce the energy consumption of the products concerned. In this global context, each machine that performs an energy conversion must be characterized by the highest possible efficiency. In the field of turbomachinery, high-performance machines have been realized over the last years thanks to the technological advancements and to the use of sophisticated design methodologies based on advanced CFD techniques. At the outset of the design process of a turbomachine, the main dimensions of the impeller (i.e. the exit diameter and width) need to be evaluated as a function of the overall required performances. For a fan, these performances include the total pressure rise, the volume flow rate and the rotational speed, which are generally expressed as non-dimensional performance parameters (i.e. pressure coefficient and flow coefficient) by means of dimensional analysis.

The preliminary calculation of the main impeller dimensions of a fan is classically performed on the basis of consolidated empirical correlations (e.g. the Cordier diagram), which relate non-dimensional performance parameters and characteristic geometric ratios to the specific speed, defined at the point of maximum efficiency of a turbomachine. The specific speed represents a parameter of great importance in selecting the most appropriate type and size of machine required for a given duty, as its value gives the designer a guide to the type of machine that will provide the normal requirement of high efficiency at the specified design conditions. Since the aforementioned literature correlations have been determined in the middle of the last century upon experimental results obtained from a large number of different type machines, these correlations need to be updated in order to design fans whose performances meet the eco-design requirements set out in the current European Regulation. In this paper, a statistical investigation on the performances and the dimensions of high efficiency existing radial flow fans is reported. Either single or double suction centrifugal fans are analyzed, with backward or forward curved blades. Some correlations between non-dimensional performance parameters and characteristic geometric ratios are found and their connection with literature correlations is pointed out. These relations can be used for a general design of high-efficiency radial flow fans.

FAN ENERGY EFFICIENCY METRICS AND FAN SELECTION IN COMMERCIAL AND INDUSTRIAL FAN SYSTEMS

Mark BUBLITZ The New York Blower Company (United States)
Tom CATANIA University of Michigan (United States)
Michael IVANOVICH AMCA International (United States)
Tim MATHSON Greenheck Fan Corporation (United States)
Nazme MOHSINA AMCA International (United States)
Trinity PERSFUL Twin City Fan & Blower (United States)
Mark STEVENS AMCA International (United States)

Fan efficiency metrics form a key component of energy efficiency regulation as applied to air movement fans. Within the United States, energy efficiency regulation has been under development but not implemented for air movement fans. Recent activity by regulatory bodies implies implementation of energy efficiency regulation is imminent in the United States. When compliance with regulation becomes legally binding the implications to manufacturers of air movement fans, because of the metric embedded in the regulation, is significant. Fan Energy Index (FEI) and Fan Electrical Power (FEP) are design-point metrics that emphasize compliant fan selections and are, as a result, a new and different approach from the traditional method of eliminating product models based on best-efficiency points and minimum efficiency thresholds. Consequently, FEI and FEP address the challenge inherent in and unique to fans: separating a fan's energy efficiency capability from the energy efficiency of the fan as applied in a system.

This paper conducts a historical review of the regulatory development in Europe and the United States, reviews and evaluates first-generation fan efficiency metrics, defines and develops the new metrics of FEI and FEP, and describes how these new metrics can be applied in regulations and rebate programs. The paper also investigates how these new metrics provide improved fan selections and as a result save more energy than traditional energy efficiency metrics. The paper concludes with a discussion on the practical consequences for air movement fan manufactures when regulation becomes legally binding in the Unites States.

THE ROLE OF UNIVERSITIES WITHIN THE QUALIFICATION AND VERIFICATION PROCESS OF EUP REQUIREMENTS FOR TURBOMACHINES USING THE EXAMPLE OF ROTODYNAMICS PUMPS

Gerhard LUDWIG Technische Universität Darmstadt (Germany)
Paul TAUBERT Technische Universität Darmstadt (Germany)
Peter F. PELZ Technische Universität Darmstadt (Germany)

The Chair of Fluid Systems has a long tradition in consulting and supporting European pump manufacturers as well as related associations in developing appropriate methods to qualify products in respect to their energy efficiencies, in order to meet the EuP requirement legislated by the European Commission.

Although it was one of the priority objectives of the developer to provide simple to use and clearly described procedures it is important to have sufficient expert knowledge as well as a suitable infrastructure to carry out profound verification tests. During the phase of product qualification, which normally is in the responsibility of the manufacturers respectively the company who places the product on the market, these skills should be available. Another situation arises in the case of verification. That means, if any market participant expresses reasonable doubt concerning the declared efficiency values and hence the market surveillance has to act. Because of the fact that the market surveillance is responsible for all issues concerning the conformity of products it is obvious that they have to do a great task, which needs a lot of expert knowledge and in many cases access to complex test facilities to carry out practical verification tests.

Besides general EuP related issues concerning the qualification approaches for rotodynamic pumps this paper demonstrates a practical example how universities can support the work of market surveillance by providing their neutral scientific expertise as well as their infrastructure in the frame of product verification. This example based on the work achieved within a small pilot project funded by the ministry of environment of Baden-Württemberg respectively the affiliated market surveillance authority. The project consists of a round robin test to verify the MEI (Minimum Efficiency Index) of a single stage end suction centrifugal pump that was taken from the market and provided by the market surveillance authority of Baden-Württemberg. The MEI value is the currently valid figure to qualify rotodynamic water pumps concerning their energy consumption. Besides the Chair of Fluid Systems three further university institutes, all with appropriate expertise in hydraulic turbomachinery, were involved in the round robin test.

The approaches concerning the support of market surveillance highlighted in this paper are not only limited to the field of hydraulic turbomachines, they can also be applied to the verification process of fans where the challenges for the market surveillance authorities are similar in the majority of cases.

FAN MODEL TEST AT VARYING AMBIENT PRESSURE: EFFICIENT PRODUCT VALIDATION AT FULL SCALE REYNOLDS AND MACH NUMBER

Sebastian SAUL Technische Universität Darmstadt (Germany)
Peter PELZ Technische Universität Darmstadt (Germany)

Common and standardized test rigs for fans operate at standard ambient conditions. An increase of Mach and Reynolds number is realized with increasing rotating speeds, but both dimensionless parameters cannot be changed individually. This is only possible, if the impeller diameter is changed, which leads to higher requirements to the manufacturing process as well as increased measurement uncertainties. The relative roughness of all surfaces and the geometry, including the relative gap, have to be scaled. When changing ambient conditions, this problem disappears. An increase in static pressure causes an increase in density and Reynolds number. The speed of sound and the Mach number are independent of the ambient pressure for ideal gases. With pressure and rotational speed as variable parameters all combinations of Mach and Reynolds numbers are possible within a specific range. The effect of the Reynolds number on performance and efficiency is examinable at constant Mach number. Compressible effects are measureable for increased Mach numbers at constant Reynolds number, because they are small for fans in comparison to Reynolds number effects.

In this paper, the physical background for a variation of ambient conditions are presented, as well as the design of the investigated pressurized test rig. This method is applicable for all types of fan models, which fit into the pressurized chamber. The limits for Mach and Reynolds number are derived, based on power and torque limits and the measurement range of all measurement techniques. The experimental investigations and the results are presented and discussed. An additional advantage of this introduced measurement procedure is the enlarged Reynolds number range without changing the fan model and test rig. Thus, the systematic measurement errors can be reduced. A variation of the gas and its properties allows a further increase in Mach number, which is necessary for a deeper investigation of compressible effects in fans or other turbomachines at subsonic operating conditions.

Summarized, this paper shows the possibilities for fan test rig operations at different ambient pressures. The advantages are the investigation of compressible effects in fans and the enlarged Reynolds number range without changing fan model or test rig.

COMPARISON OF LIGHTHILL'S ANALOGY AND ACOUSTIC PERTURBATION EQUATIONS FOR THE PREDICTION OF HVAC BLOWER NOISE

Matthias TAUTZ Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Manfred KALTENBACHER Vienna University of Technology (Austria)
Stefan BECKER Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

The prediction of noise emitted from rotating turbomachineries is a major concern in many industries. Products that directly affect the comfort of costumers are of particular interest. In this context, our aim is to develop reliable tools for the aeroacoustic noise simulation of radial blowers in heating, ventilation and air conditioning (HVAC) systems. We use a hybrid, volume discretized approach for this purpose. As the first step, the flow field is computed by applying the commercial finite volume solver Star-CCM+, where the WALE large eddy simulation (LES) turbulence model is used and constant density is assumed. This reduces the computational cost to receive a non-stationary flow field. As a consequence thereof, no acoustic information is incorporated in the flow pressure. Subsequently, aeroacoustic sources are evaluated on the flow mesh and conservatively interpolated onto a coarse finite element mesh. Finally, the in-house code CFS++ is applied for computing the sound propagation by using a non-homogeneous wave equations.

This simulation procedure is rather complex because two volume discrete meshes need to be created, which are furthermore split into a static and a rotating domain. Its major advantage is the full insight into the acoustic source distribution and propagation field. The acoustic perturbation equations (APE) are used in form of the perturbed convective wave equation (PCWE) and compared to the Lighthill analogy. The optimal simulation configuration is determined for each method by variation of the CFD mesh size and the spatial extent of the recognized sources. Furthermore, their implementation challenges and details are outlined. While both approaches yield sound pressure spectra that are in good agreement with experimental data, the PCWE has the advantage of separating flow and acoustic pressure, thus providing more information on the sound excitation processes in the blower. Its drawback is the high sensitivy to numerical noise of the flow simulation pressure. Taken together, the presented methologies can be used in radial blower development with respect to noise emission.

INTEGRATED CFD-ACOUSTIC APPROACH TO THE SIMULATION OF TONAL AND BROADBAND NOISE GENERATED BY AXIAL AND CENTRIFUGAL FANS

Piergiorgio FERRANTE NUMECA International (Belgium)
Paolo DI FRANCESCANTONIO STS - Scientific and Technical Software (Italy)
Charles HIRSCH NUMECA International (Belgium)

An innovative computational approach for the simulation of noise generated by axial and centrifugal fans is presented, integrating mesh generation, CFD simulation and acoustic propagation analysis. The simulation chain includes methods for the efficient and accurate simulation of tonal and broadband noise, providing affordable alternatives to more traditional but expensive unsteady approaches.

The simulation of tonal noise is based on the application of the Non-Linear Harmonic (NLH) method for the noise sources related to blade passing frequencies and their harmonics, as well as propagation in the fan duct (including sound absorbing material) and in the near-field, followed by a propagation analysis to the far-field based on either a Boundary Element Method (BEM) or a Ffowcs Williams - Hawkings (FW-H) formulation. With this approach the simulation time is one to two orders of magnitude faster than with full-unsteady traditional multistage approaches with URANS models.

Typical computational approaches for broadband noise are based on time domain methods (e.g. LES, DES) requiring large computational efforts. The cost-efficient approach proposed by NUMECA exploits the Adaptive Spectral Reconstruction (ASR) method, based on a stochastic reconstruction of the turbulent noise sources. The method relies on a steady RANS solution allowing the application for preliminary design assessment and optimization. The frequency content of the turbulent field and the spatial cross-correlation of the turbulence scales are reconstructed. The noise sources (Lamb vectors) are extracted from the reconstructed turbulence and propagated in the near-field with a Finite Element Method (FEM) solving the Pierce-Howe Convective Wave Equation in the frequency domain. The far-field radiation is finally computed by a BEM solver.

The above mentioned computational chain is implemented in NUMECA's Flow Integrated Environment (FINE) suite, providing wizard automation and best practices. In the frame of this study, the suite is applied to the simulation of the noise generated by two different axial fans, respectively the Advanced Noise Control Fan (ANCF) operated by NASA Glenn Research Center and the NPU-fan, currently exploited in the IMAGE project, operated in the Turbomachinery Aerodynamics and Acoustics Laboratory (TAAL) of the Northwestern Polytechnical University of Xi'an, China. Centrifugal fans will be also considered. For all the fans, tonal and broadband noise predictions are compared with experimental data in order to demonstrate the level of accuracy and the computational performance. The results obtained indicate in general a reasonable prediction of tonal and broadband noise in terms of levels and trends with some exceptions for specific propagation angles.

IMPROVED ANALYTICAL PREDICTION OF BOUNDARY LAYER INDUCED ROTOR NOISE USING CIRCUMFERENTIAL MODES

Martin STAGGAT German Aerospace Center (DLR) (Germany)
Antoine MOREAU German Aerospace Center (DLR) (Germany)
Sébastien GUÉRIN German Aerospace Center (DLR) (Germany)

Modern aircrafts have to meet strict regulations concerning the emission of pollutants and noise to limit the environmental impact and ensure the acceptance of growing air travel. This demand will not be satisfied by the classical tube-wing aircraft architecture, as it has already come close to its maximum of efficiency. The introduction of revolutionary technologies, such as boundary layer ingesting engines, has the potential to bridge the gap towards a transformation of aircraft shape to highly fuel efficient blended or hybrid wing bodies with embedded engines. Beside the potential benefits of a boundary layer ingesting engine concept new concerns arise related to the noise emission of such configurations resulting from the highly distorted engine inflow.

The DLR department of engine acoustics is developing an analytical tool that allows the modular prediction of several tonal and broadband noise sources. In an earlier publication an analytical theory, based on a modal approach in the frequency domain, was presented for the prediction of broadband noise caused by the ingestion of a turbulent boundary layer. This theory is able to predict peaks in the broadband noise spectrum at the blade passing frequency and its higher harmonics for various operating conditions of the rotor

The current work aims on the improvement of this theory. The prior used one-dimensional isotropic von Kármán spectrum is replaced by a one-dimensional Kerschen-Gliebe spectrum to account for the anisotropy of the inflow. The estimation of the transversal coherence length is substituted by a recently published model that was specifically developed for a turbulent boundary layer. This model replaces an ad-hoc assumption made for the axial coherence length, which determines the intensity of blade-to-blade correlation and is therefore a steering parameter.

To verify the adapted theory, noise predictions are compared with acoustic measurements of the "Fundamental Test Case 3 (FC3)" published in the framework of the "*Fan Broadband Noise Prediction Workshop 2015-17*".

APPLICATION OF A DISCONTINUOUS GALERKIN BASED CAA SOLVER FOR BROADBAND NOISE PREDICTION

Lev LIBERSON German Aerospace Center (DLR) (Germany)
Markus LUMMER German Aerospace Center (DLR) (Germany)
Michael MÖSSNER German Aerospace Center (DLR) (Germany)
Roland EWERT German Aerospace Center (DLR) (Germany)
Jan Werner DELFS German Aerospace Center (DLR) (Germany)

Computational simulation plays a significant role in the development process of modern industrial fans. Computational fluid dynamics (CFD) is widely used for aerodynamic optimization and to gain a fan design with improved efficiency. In addition, the acoustic performance becomes more and more important as it constitutes another unique selling proposition. To obtain a low-noise fan design, efficient computational methods are necessary for the prediction of aerodynamically generated noise. The development of a first principle based broadband fan noise prediction capability is the aim of a cooperation between the industrial partner ebm-papst and the department of technical acoustics of the German Aerospace Center (DLR). A recently developed computational aeroacoustics (CAA) solver DISCO++ is used for broadband noise assessment utilizing stochastically generated sources.

The mechanism of noise generation by the flow field of an axial fan is a highly complex three dimensional process. A prominent example is the interaction between blade-tip vortices with the surrounding tip geometry and trailing edge noise from the fan blades. An efficient option for spatial discretization of such highly complex geometric features is provided by unstructured tetrahedral meshes. Such an approach is used in the CAA code DISCO++ to solve the acoustic perturbation equations (APE) with the discontinuous Galerkin (DG) method.

To provide a method with comparatively manageable numerical effort a two-step hybrid approach is chosen. In a first step, a Reynolds Averaged Navier Stokes (RANS) simulation is performed to obtain the stationary flow field and statistical turbulence parameters. Thereupon, time-resolved synthetic turbulence is generated by the Fast Random Particle Mesh (FRPM) method on a separate Cartesian grid. From the reconstructed stochastic turbulence an acoustic source term is computed and passed to DISCO++ which in turn computes the sound propagation in the CAA domain.

Within this work the results of the DISCO++ simulations of a five bladed ducted axial fan using an acoustic source term based on vorticity structures are presented. The results are compared with experimentally obtained reference data. A good agreement over a wide range of frequencies is achieved. It is believed that a vorticity based acoustic source provides the capability to model anisotropic effects of vortex stretching in areas of highly accelerated flows occurring in the tip gap area.

EVALUATION OF ANALYTICAL AEROACOUSTIC MODELS FOR A LOW-SPEED AXIAL VENTILATION SYSTEM

Marlène SANJOSÉ Université de Sherbrooke (Canada)
Miguel PESTANA Université de Sherbrooke / Ecole Centrale Lyon (Canada)
Stéphane MOREAU Université de Sherbrooke (Canada)
Patrice CAULE Safran Ventilation Systems (France)

Several analytical aeroacoustic models are applied for a compact ducted ventilation system with high rotational speed with uniformly distributed blades and vanes. RANS simulations are performed at several mass-flow rates and provide the parameters to calibrate the noise sources for the rotor and stator acoustic models. Several blade response and acoustic propagation models have been evaluated. Due to the limited numbers of blades and vanes, the cascade effects are shown to be negligible. The propagation assuming infinite duct allows capturing the cut-off frequencies but does not modify the global spectral shape of the results. The flow parameters which define the strength of interaction mechanisms are the crucial parameters for the acoustic results. For tonal noise, the rotor wake deficit at the stator leading edge can be modeled using a Gaussian shape evolution model fitted from RANS extracted data. This model provides a satisfactory estimation and a good trend for flow rates above design but is not robust for the lowest flow rates investigated. The third blade passing frequency is overestimated by 8 dB with the present wake evolution model and with an isolated airfoil response with in-duct propagation. For broadband noise, turbulence ingestion by the rotor, turbulent rotor wake interaction with the stator and scattering of the turbulent eddies in the boundary layer at the trailing-edges of the rotor and the stator rows are investigated. The latter mechanism on the rotor is found to be dominant as the high rotational speed of the machine and the strong loadings induce high shear stress and pressure gradient on the blade surfaces. The global trend of the acoustic power evaluations with mass flow variation for the broadband noise levels estimated from the four contributions agree with current experimental investigations. For Strouhal numbers, based on fan diameter and tip velocity, above 3, the broadband noise spectrum is predicted within 3 dB accuracy with isolated response and free-field propagation.

MAXIMAL ACHIEVABLE EFFICIENCY OF FANS

Konrad BAMBERGER University of Siegen (Germany)
Thomas CAROLUS University of Siegen (Germany)

Due to increasing energy cost and the challenges in the context of climate change there is a permanent demand to enhance the energy efficiency of fans. This paper discusses the theoretical aerodynamic efficiency limit that cannot be exceeded regardless of the effort made to optimize the fan. It is distinguished between two efficiency definitions (total-to-total and total-to-static) and four fan types (axial rotor-only, axial with guide vanes, radial rotor-only and radial with volute). For each fan type, the inevitable aero-dynamic losses are estimated as a function of the design point and the Reynolds num-ber. Inevitable losses are e.g. friction losses, shock losses and exit losses. Aiming at the insuperable efficiency limit, the models to estimate the friction losses are based on a set of idealizing assumptions and the exit losses are minimized by an optimal spanwise load distribution. Since the focus is on the aerodynamic efficiency limit, losses in the motor and the drive drain are neglected.

The resulting efficiencies are depicted assuming an exemplary Reynolds number of one million. It is found that the impact of the design point is very strong, especially with re-gard to the exit losses which increase with decreasing specific fan speed and diameter. Friction losses become relevant at design points with high pressure coefficients. At such design points, the width of radial impellers becomes very small and axial fans feature large hub-to-tip ratios wherefore the wall effects from hub and shroud increase. The ef-ficiency of radial fans is further impaired by increased friction between the bottom disc and shroud with the surrounding air.

CONSIDERATIONS CONCERNING THE POSSIBLE FAN EFFICIENCY IN REAL LIFE APPLICATIONS WITH LIMITED FAN SIZE

Frieder LÖRCHER Ziehl-Abegg SE (Germany)
Walter ANGELIS Ziehl-Abegg SE (Germany)

For a large number of fan demands with limitation of the fan size, the specifications have been analyzed. The considered fan demands relate to axial, backward- and forward curved centrifugal fans and mixed flow fans, with or without guide vanes or housings. However, the fan type is often not crucial for a fan demand; to a greater degree, achievable efficiency and/or noise level are important and the choice of fan type is subordinated and can be adapted to the needs.

The requirements are separated in two groups, first, the "basic" requirements which are duty point and fan size, and which are easy to fulfil but essential for the working task of the fan. Secondly, we have "quality" requirements as low rotation speed, low energy consumption or low noise level which result of a good fan choice and a high degree of optimization.

From the basic requirements, a non-dimensional parameter δ^* is deduced which is a constraint for the fan provider or designer, and which cannot be bypassed. δ^* is a binding link between fan application and non-dimensional fan design, however it is not constant for a given fan but depends on the resistance curve at which the fan is operated. δ^* has a strong relationship to the specific fan diameter δ, resulting in the fact that the achievable quality, especially efficiency and noise, depend on this non-dimensional parameter δ^*.

The efficiency and noise of a fan depend on the value of δ^* at which it is operated. This may differ significantly from a region of values of δ^* where the fan has good efficiencies. Thus the maximum efficiency of a fan, often used as quality measure, is in general not an indicator of its energetic efficiency in a given application with size limitation.

Illustrated by typical examples, forms of fan size limitations, such as installation space, available drive torque or impeller mass, are presented and discussed. The approach of extending the fan size within a limited installation space in order to achieve higher efficiencies is addressed for a typical centrifugal fan case.

We further present a strategy of clustering the δ^* values to a limited number of regions, which can be covered each obtaining good efficiency by one non-dimensional fan design, limiting the number of required fan designs. For some of the clusters, fan designs are characterized, ending up in different axial, mixed flow and centrifugal fan designs.

DESIGN STUDY TO AXIAL FLOW FAN STATOR BLADE ROW USED IN REVERSE VENTILATION

Vaclav CYRUS AHT Energetika Ltd. (Czech Republic)
Jan CYRUS AHT Energetika Ltd. (Czech Republic)

Fan flow directional change is frequently required in emergency situations in traffic tunnels, chemical plants and mine ventilation systems. Our paper focuses on the reversing method, namely changing direction of the fan rotation and turning of the stator vanes. The paper will discuss the aerodynamic design of stator blades of two reverse ventilation axial fan stages A and B with different stage parameters, specifically the flow coefficients of 0.40 and 0.35 and pressure coefficients of 0.30 and 0.42. Free vortex blading was used when designing both fan stages.

Companies which procure reverse axial fans request that the one-shot stator vanes will have the capability of rotating around their axis in emergency events to enable the flow reversal. Consequently, the blade cascades have the higher pitch/chord ratio than one along the full blade span. New stator vane shape for the reverse fan is characterised by the decrease in the chord length in the radial direction to the fan axis. Thus the cascade aerodynamic loading increases in this region in comparison with the original stage design.

This is only possible if the aerodynamic loading of blade cascades is moderate. The cascade aerodynamic loading is often expressed by the Lieblein's definition of the diffusion factor D. If the diffusion factor is higher than 0.6, the flow is separated on the cascade profile suction surface. The paper also presents derived relationships to determine the fan stage parameters area, the pressure and flow coefficients, where the stator vane cascades would not incur the flow separation.

Following the analysis of the stage A we conclude that using the stator vanes for the flow reversal is acceptable. The maximum diffusion factor at the stage hub is 0.56. However, in analysing the fan stage B we find that the stator diffusion factor is above the critical value of 0.6 in lower half of vane height near the fan hub. This condition will cause large flow separation area together with unacceptable decrease in the fan efficiency.

For that reason the authors designed the stator vanes consisting of two parts, straight which is fixed and curved which is moveable. At the basic flow conditions these parts are connected. During the flow reversal the moveable curved stator blade part is rotated to ensure the maximum flow rate.

Computed (CFD) and measured aerodynamic performance of the original stage B with profile blades was compared resulting in acceptable agreement for the basic flow direction. As a result it was possible to investigate the effect of reversing the one-shot stator vane design variants on the fan stage efficiency with the use of CFD data. The decrease in efficiency at the design conditions was 0,5 % to 1,2 % in both studied stages.

INVESTIGATION ON THE MAXIMUM TOTAL TO STATIC EFFICIENCY OF AXIAL FANS WITH AND WITHOUT DIFFUSERS

Johannes WALTER Karlsruhe Institute of Technology (Germany)
Saban CAGLAR Karlsruhe Institute of Technology (Germany)
Martin GABI Karlsruhe Institute of Technology (Germany)

The range of kinetic energy at the outlet of a fan depends on the fan design and the operation point. In most configurations the kinetic energy at the outlet is a loss and thus affects the fan efficiency. Diffusers can be installed to reduce these losses. In this study the maximum static efficiency

related to the total efficiency is analyzed for a fan with and without outlet guide vanes. In order to achieve a further efficiency increase, the effect of a diffuser is studied. The kinetic energy can be divided into two components, a meridional and a circumferential component. Both components depend on the hub-to-tip ratio and the swirl distribution. In this study the common free vortex design is considered and the hub-to-tip ratio is chosen according to the Strscheletzky criteria. In the first part, the maximum static efficiency of fans with and without outlet guide vanes is analyzed. The outlet guide vanes lead to the elimination of swirl and thus increase the efficiency. A further possibility to increase the efficiency is the installation of an annular diffuser

downstream the axial fan. The diffuser reduces the kinetic energy at the outlet and thus the Carnot losses. A large reduction of the kinetic energy is achieved by a big area ratio of the diffuser. Since the opening angle of the diffuser is limited this can lead to long components. There is a correlation

between area ratio, length and losses in the diffuser. The static efficiency is analyzed for a diffuser length which corresponds to the outer fan diameter. There is a significant increase compared to the latter configurations.

PROFILES FOR REVERSIBLE AXIAL FANS

Georg KLEPP HS-OWL (Germany)
Markus FILIPPI HS-OWL (Germany)

Most fans have a distinct flow direction, nevertheless a reversible flow direction for fans is desirable in certain circumstances in ventilation and drying applications. In order for the fan performance to be similar in both directions, the blade profiles of these axial fans have to conform to distinct symmetry conditions. As data for this kind of profiles is very scarce, the flow around such reversible profiles is investigated. Thus it is possible to get further insight into the aerodynamics in order to improve the efficiency of the profile design.

Most common are elliptical airfoils and S-shaped profiles. These are mostly derived from existing profiles using a twice cambered (i.e. S-shaped) camber-line.

The flow over elliptical airfoils and S-shaped profiles is investigated numerically solving the two dimensional incompressible Reynolds averaged Navier-Stokes (RANS) equations. The influence of different turbulence models is shown. The results are validated and compared with experimental data from the literature. A comparison with data obtained using other computational methods is performed.

The flow around various profiles at various incident angles and for various low Reynolds numbers, as appropriate for fan profile applications, is computed. The flow patterns of the flow around the profiles are investigated and polar diagrams are derived from the computations. The performance of the S-shaped profiles is compared to the elliptical airfoils and the normal (once cambered) airfoils.

With an increase in camber and profile thickness lift and drag also increases. The most important parameter is the camber. The thickness distribution has a smaller influence. Due to the S-shape the flow is asymmetric and complex: There is always flow separation as well as reattachment and changes in the direction of the pressure gradient due to the curvature changes. For strong cambered profiles, these effects are predominant and the advantages using a twice cambered profile might diminish.

A CENTRIFUGAL FAN TEST BENCH FOR VALIDATION DATA AT OFF-DESIGN CONDITIONS

Balazs PRITZ Karlsruhe Institute of Technology (Germany)
Johannes WALTER Karlsruhe Institute of Technology (Germany)
Martin GABI Karlsruhe Institute of Technology (Germany)

Nowadays with eco directives becoming more strictly the efficiency improvement of turbomachinery is getting more important than ever. Relieving experimental investigations computational fluid dynamics (CFD) could be the major tool to optimize the geometry of a turbomachinery, as it can provide a detailed insight into the flow within the impeller. In order to have reliable results from CFD simulations a validation of the acquired simulation tool and the composed numerical model is crucial. Especially in off-design conditions it is very challenging for CFD to predict strongly whirling, highly unsteady flow fields correctly.

In the present paper the construction of a test bench for a centrifugal fan with spiral casing is described and the first results are presented. The main goal in constructing the test case is to provide extensive validation data not only in the design operation point but for off-design conditions also. The geometry was designed based on classical design guidelines. It is close to industrial centrifugal fans however an additional aim was, to provide a geometry which allows an acceptable mesh generation effort and high mesh quality. Due to the rotational periodicity of axial and radial fans without casing, those types can be calculated by regarding only a single blade passage, which decreases the number of required cells and hence the necessary computational power. For centrifugal fans with spiral casing this simplification is not suitable and this construct induces additional issues as pressure pulsation, acoustics, pulsating radial force, periodic load of vanes etc. The present test case should provide availability to investigate such phenomena also.

The characteristic and the efficiency curve of the fan are measured with standardized methods. Beyond this "Particle Image Velocimetry" (PIV) is chosen to measure the flow field within the blade channels and the spiral casing, as well, in several different positions relative to the corner ("tongue"). For the optical accessibility the hub of the impeller and the main parts of the casing are made out of acrylic glass. In order to keep the seeding particles within the system a closed loop test rig is constructed. The inlet section of the fan was designed to get well defined inlet conditions for the numerical simulation.

The first investigations were carried out at a rotational speed of 600 rpm, and compared with results from CFD. These results are presented and discussed in the present paper. Further investigations will be performed at higher rotational speed.

END-OF-LINE BEARING QUALITY MONITORING ON COMPACT VENTILATORS: A CASE STUDY

Julien GRILLIAT ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
Jérôme ANTONI INSA de Lyon (LVA) (France)
Michael D. COATS University of New South Wales alumni (Australia)

In the market segment of decentralized domestic ventilation, compact blowers and axial fans are utilized. The fan acoustic performances are thereby a very important sale argument. Indeed, these fans are likely to be installed in bedrooms and disturb the customer's sleep. When running at very low rotational speed (so called "sleep mode"), airborne noise is negligible whereas structure borne noise is very important. Intern preliminary studies have shown that damaged ball bearings can enhance the fan acoustic emissions up to 8-9 dB(A). A bearing monitoring in End-of-Line is therefore of greatest importance.

The present case study will be divided into the following steps:

Problem description: end-of-line testing has to cope with different constraints. Some are technical (background noise, hardware), others are economical (production cycle time). The full framework will be presented.

Technical main solution: bearing monitoring is carried out by means of vibro-acoustic measurements envelope analysis with peak mode, inspired by commercial codes. This method will be briefly introduced and its choice will be justified. The method's advantages and drawbacks as well as its practical requirements and possible conflicts with the aforementioned constraints will be scrutinized.

Practical implementation: this part will be the presentation core. A step-by-step description of the implementation will be presented in order to present the difficulties the authors had to cope with. The main technical points include sensor positioning and fan rotation rate correction. Indeed, the fan driving electronics do not allow a very stable rotation rate, which though is one of the key points in order to get a monitoring with good quality. In order to compensate rotation rate fluctuations, order tracking had to be implemented. Several techniques were tested. A last point deals with automatic decision and quality management. On one hand, operator influence shall be reduced to its minimum. On the other hand, the process quality has to be ensured, which is often evaluated with repetition tests. This represents a great challenge, since peak mode analysis does not allow repetitive results.

Interest of the present communication: End-of-line Testing and condition monitoring do share the same methods and face the same challenges when trying to extract information from vibroacoustic measurements.

INTEGRATED MANAGEMENT OF EXPERIMENTAL RESEARCH- AND META-DATA FOR FAN TEST RIGS

Nils PREUSS Technische Universität Darmstadt (Germany)
Peter F. PELZ Technische Universität Darmstadt (Germany)

The availability of descriptive metadata is mandatory for the long term usability of experimental research data. This paper introduces a modular data model based on the HDF5 file format that is applicable for the typically heterogeneous and evolving experimental setups. It enables uniform storage of related measurement data in different processing stages and corresponding machine actionable metadata in the same file. On this basis a software design is proposed that streamlines data storage, access and processing by means of object oriented programming, limiting the need for customized data acquisition and processing software per test rig configuration. Furthermore, application examples for typical fan test rigs are presented and discussed.

INVESTIGATIONS ON NOISE SOURCES ON A CONTRA-ROTATING AXIAL FAN WITH DIFFERENT MODIFICATIONS

Ralph KRAUSE Institut für Luft- und Kältetechnik gGmbH (Germany)
Christian FRIEBE Institut für Luft- und Kältetechnik gGmbH (Germany)
Michael KERSCHER gfai tech GmbH (Germany)
Christof PUHLE GFaI e.V. (Germany)

Axial blowing fans are commonly known in many fields of applications. By using axial fans swirls occur at the trailing edge of the blades due to the working principle. These swirls are unwanted in most cases and may have unfavourable influence on subsequent devices, e.g. higher pressure drop or lower heat transfer coefficient. As the static pressure rise is an evaluation criterion for the fan efficiency there are different possibilities for converting the dynamic pressure of the swirl into a static pressure rise. The most common application for rising the efficiency is the installation of discharge guide vanes.

The installation of contra-rotating axial fans (CRF) is another well-known opportunity to increase the efficiency of a fan. Besides the high power density and the high efficiency it should be noted that these fans are known for problematic noise behaviour. This is induced by the interaction of both wheels with different directions of rotation.

An Acoustic Camera was applied to examine various modifications of fan blade designs regarding their noise emissions. A so-called rotational beamforming algorithm allows for the detection of sources on the rotating blades by using a virtual rotation of the microphones. Depending upon the frequency different sources could be localised.

Both the leading and trailing edge were modified. The leading edge was shaped with a sinusoidal structure and shall lead to a lower leading edge noise, which is mostly induced by impinging vortices. The trailing edge was modified using serrations. These serrations shall lead to vortices at higher frequencies.

This paper shows the performed modifications and tests with the Acoustic Camera. It also presents first results and gives an outlook on future work and usage.

INFLUENCE OF BLADE SWEEP ON AERODYNAMICS AND ACOUSTICS OF LOW-PRESSURE AXIAL FANS

Alexander SKORPEL Technische Universität Braunschweig (Germany)
Philip FRANTZHELD Technische Universität Braunschweig (Germany)
Jens FRIEDRICHS Technische Universität Braunschweig (Germany)

In this paper three different low-pressure axial fans are experimentally and numerically investigated. All fans are following the same design process using different values to influence the degree of blade sweep being applied. In previous investigations forward sweep was applied to successfully reduce sound emissions. Observations showed: the larger the maximum sweep angle the larger the reduction of noise. The reduction of the section lift coefficient was compensated by an increase in chord length accepting a small loss in efficiency. The observed effect is now being investigated in a more intensive measurement program focusing on tip clearance flow.

Fan No. 1 is considered as a reference fan. Fans No. 2 and 3 differ in the shape of the radial sweep angle curve; No. 2 reaching for a very extreme value at blade tip, No. 3 reaching for the same value at blade tip as the reference but with a bulkier curve shape at blade mid-height.

All fans are experimentally investigated in an on-site fan test chamber. In addition to basic fan characteristic measurements the wake flow field is measured using a five hole probe. For further investigations on the tip vortex phase-locked wall pressure measurements are carried out. These measurements are then used to validate the numerical RANS-simulations that give a more detailed view on the complex three-dimensional flow phenomena near the blade tip. Externally conducted acoustic measurements show the sound power level on suction and on pressure side as well at different operation conditions. The attempt is made to find a correlation between used design parameters, tip-clearance flow field and the effect on overall sound emission.

TONAL NOISE PREDICTION OF UNEVENLY-SPACED BLADES AXIAL FANS BASED ON BLADE FORCE MODEL

Yadong WU Shanghai Jiao Tong University (China)
Dinghao PAN Shanghai Jiao Tong University (China)
Zhigang PENG Shanghai Jiao Tong University (China)
Hua OUYANG Shanghai Jiao Tong University (China)

Tonal noise is the main noise source of the automotive engine cooling fans. Many methods are adopted to reduce the tonal noise level, such as uneven-spaced fan, swept and leaned fan. The uneven-spaced fan technique is widely used in the automotive engine cooling fans. As normal, the uneven-spaced fan is designed applying phase modulation principle according to adjusting the blade spacing angles, that is the Modulated Blade Spacing method. For that method, the blade forces were not taken into account, and the blade forces have much effect to the noise prediction. In this paper, a detailed investigation was conducted on the transient characteristics of blade forces within uneven-spaced fans.

To obtain the blade force, CFD modeling of two axial low-speed fans, one unevenly-spaced and another evenly-spaced were carried out. Discrepancies and characteristics in blade-forces' within uneven fans were identified and thus modeled. The blade segmentation strategy was used to abstract the blade force from unsteady Reynolds-averaged Naviers Stocks simulation, propagated into the far field using point-force model derived from Lowson model. Then the noise modulation relation was set up based on the blade spacing angle and blade force. Verifications are done with comparing results with experimental data, Duncun model, as well as the prediction done by applying the Lowson model directly to the calculated forced items get from CFD. The comparison results show that the modified noise modulation prediction model agreed well with the experiment.

3D INVERSE DESIGN BASED OPTIMIZATION OF MULTI-BLADE ROW AXIAL FANS USED FOR DISTRIBUTED PROPULSION

Mehrdad ZANGENEH University College London (United Kingdom)
Peng WANG Advanced Design Technology Ltd (United Kingdom)
Pedro GOMES Advanced Design Technology Ltd (United Kingdom)

In many fan applications such as electronic cooling or tunnel ventilation rotor/stator or contra-rotating fans are used. In many cases because of packaging limitations the blade rows have to operate with quite small axial spacing and hence there is strong interaction between the blade rows. It is hence important to be able to design and optimize these fans taking account of the strong interactions between blade rows.

In this paper, a new methodology is presented for design of multi-blade row fans where the blade rows are designed by using a viscous 3D inverse design method which takes account of the strong interaction between blade rows. In this inverse design method the blade geometry is computed for a specified distribution of blade loading subject to a specified spanwise work distribution. The unique feature of the method is to combine a 3D viscous inverse design method with a robust mixing plane method satisfying interface flux conservation, non-reflectivity and retaining interface flow variation. The method can be used in all Mach numbers by using a low Mach number preconditioning technique.

In this paper the methodology used is presented and then applied to a rotor/stator axial fan application. The method is used to design the fan rotor both in stage mode (rotor/stator) and in single blade row mode by using the same blade loading for the rotor. Hence only the rotor geometry is changed and the stator geometry is not changed from the baseline design. The flow field in the resulting stages (New rotor designed in stage mode with baseline stator and the new rotor designed in single blade row mode with baseline stator) are then analysed by using commercial CFD code CFX and significant improvement in stage efficiency is observed for the stage design.

STRUCTURE AND KINEMATICS OF THE VORTEX SYSTEM IN AXIAL TURBOMACHINES

Paul TAUBERT Technische Universität Darmstadt (Germany)
Peter F. PELZ Technische Universität Darmstadt (Germany)

This investigation models the kinematics of the vortex system of an encased axial turbomachine depending on the operation point applying vortex theory and potential flow theory. By doing so, the physical understanding of compressor and fan stall at the housing, surge, spill forward and noise generation is enlarged. The total change in circulation along the plane of the machine is linked to the flow number $\varphi := U / (\Omega a)$ by Euler's turbine equation, with U the axial free-stream velocity, Ω the angular frequency of the turbomachine and a the radius of the housing. Depending on the operating point, the vortex system of an axial turbomachine changes. For the nominal operating point $\varphi \approx \varphi_{opt}$ and negligible induction, the tip vortices transform into a screw with a pitch of $2\pi\varphi a$. The most relevant operating point for practical applications is part load operation. For this case, i.e. small flow numbers $\varphi \to 0$, the tip vortices roll up to a vortex ring with increasing circulation due to Helmholtz's theorems. The vortex ring itself is generated by bound vortices rotating with the angular frequency Ω.

Two results emerge out of the analytic research concerning the vortex system of an axial turbomachine at part load: first, the analysis shows an induced movement of the tip vortices in the same rotating direction as the turbomachine. The vortex ring rotates at the sub-synchronous frequency $\Omega_{ind} < \Omega/2$. Second, the vortex ring itself induces an axial velocity at the tube wall. Superimposed with the axial main flow, this results in a stagnation point. Since the vortex strength increases in time, the stagnation point moves upstream. This effect may falsely be interpreted as a dynamic boundary layer separation. Hence, the results may give new insights into transient stall phenomena in axial turbomachinery. For overload $\varphi \to \infty$ the hub, the bound and the tip vortices form a horseshoe. Both, hub and tip vortices are semi-infinite, straight vortex filaments generated by bound vortices. The analysis yields an induced movement of the tip vortices against the rotating direction of the turbomachine at heavy overload.

EXTENDED SCALING METHOD FOR NONSIMILARITY IN REYNOLDS NUMBER, STAGGER ANGLE AND BLADE NUMBER

Sebastian SAUL Technische Universität Darmstadt (Germany)
Peter PELZ Technische Universität Darmstadt (Germany)

The similarity of Reynolds and Mach number, relative roughness and relative gap for fan model and prototype cannot be retained for the dimensioning of large industrial fans. Thus the efficiency of model and prototype changes. Scaling methods are supposed to compensate these differences in efficiency from model to prototype.

The efficiency $\eta=\eta(\sigma,Re,Ma,k_+,s_+,\varphi)$ depends on the type (specific speed σ), the dimensionless size (Reynolds- and Mach number Re,Ma), the quality (relative roughness and relative gap k_+,s_+) and the operating point (flow coefficient φ). The change of stagger angle results in a change of specific speed or fan type. The loss distribution changes and the scaling method shall include that change. Common scaling laws like Ackeret's formula are easy to use, but they show a large deviation between prediction and measurement. This paper focuses on incidence losses for axial fans. The loss model and the limits are an important part of the paper, which is an expansion of the scaling method Pelz & Saul (2017). The predicted efficiencies are validated by experimental investigations of a fan model with a specific speed $\sigma=1…1.4$ with a stagger angle range of $\Delta\beta_s=-18°:6°:+12°$. Additionally, measurements are done for full- and half-bladed rotor, which is a common method of the industry to match a fan to a specific task. Measurements show, that the incidence angle and the number of rotor blades have an impact on the efficiency and the efficiency scaling potential, which is $\Delta\eta=3…5.5$ % in the Reynolds number range of $Re=2.2…6.5$ E6. The presented scaling method considers the incidence angle with the help of design and flow parameters, which are all known parameters.

The result is an extended scaling method, taking incidence loss into account.

PSYCHOACOUSTICAL INVESTIGATION OF SOUNDS FROM HEAT PUMPS

Carolin FELDMANN University of Siegen (Germany)
Thomas CAROLUS University of Siegen (Germany)
Marc SCHNEIDER ebm-papst Mulfingen GmbH (Germany)

Heat pumps become increasingly popular as heating systems for houses. Purchasing a heat pump or replacing the conventional heating system in favor of a heat pump is mainly driven by a growing ecological sensibility, costs for energy and legal requirements.

The air delivering fan is an essential component in an air-to-air heat pump and simultaneously the main acoustic source. Typically, air-to-air heat pumps are placed outside of the house, and hence a source of annoyance for the neighborhood. According to the German regulations the maximum rating level is 55 dB(A) in residential areas by day and 40 dB(A) at night. Although this rating level already contains A-weighting and penalties for tonality and impulsiveness, the degree of annoyance of different heat pumps with equal rating levels is assessed as completely different. Humming for instance is a frequent reason for complains.

The aim of the current study is to evaluate subjective response from jury tests to objective metrics such as loudness (DIN 45631, ISO 532), sharpness (DIN 45692), tonality (DIN 45681, ISO 1996-2) and others.

Different sounds of heat pumps were evaluated by a jury of in total 40 participants. The selected sounds were adjusted to the same A-weighted overall sound pressure level to determine level independent influences as for example tonality and impulsiveness used for the rating level.

The test was based on seven-point adjective scales for sound description and assessment. In addition the test persons were interviewed concerning their sound preferences. Based on the jury test results the sound samples and the adjective scales were grouped. The results show no correlation to the rating level at all, e.g. no correlation to possible tonality or impulsiveness. Furthermore the location of the maximum spectral energy is important for heat pump sounds avoiding the frequency range that generates a sensation of "humming" as well as "whistling" or "hissing". In addition, the time structure of the sound plays an enormous role for the perceived pleasantness or to blank out the sounds which could not be represented by the impulsiveness used in the rating level. Eventually, a preliminary regression model for sound quality is presented. It is proven that - beyond the standard rating level - psychoacoustic parameters are suitable for sound quality assessment of heat pumps.

THE RELATIONSHIP BETWEEN PERCEPTUAL DIMENSIONS OF FAN NOISE AND PATTERNS OF THE SPECIFIC LOUDNESS

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

Fan noise is a typical component of environmental noise perceived by humans in every day situations. In order to enable a successful development of more pleasant fan sounds, it is desirable to understand and characterize the perceptually relevant aspects of fan noise beyond current technical measures like the dB(A).

The aim of this study is a determination of the perceptual dimensions of fan noise, an identification of the dimensions which are relevant for the (un-)pleasantness of fan noise and the development of (psycho-) acoustic parameters reflecting the most relevant perceptual dimensions. In a listening experiment, 35 different fan noises were rated by 45 participants using a semantic differential. The semantic differential consisted of 29 adjective scales covering evaluative items (like e.g. unpleasant - pleasant) and also descriptive items (like e.g. not humming - humming).

Based on two different factorial analyses of the fan noise ratings, six perceptual dimensions and five groups of sounds could be identified. The six perceptual dimensions describe how (I) pleasant, (II) humming/bass, (III) shrill, (IV) monotone, (V) reverberant and (VI) noise-like the fan sounds are. The five groups of sounds can be characterized as (A) unpleasant, (B) humming, (C) pleasant, (D) noise-like and (E) varied. It turns out that the pleasant and the unpleasant sounds can be mainly distinguished based on the first three perceptual dimensions. The group of pleasant sounds (C) is middling on the factor humming/bass and is not shrill, whereas the group of unpleasant sounds (A) is slightly humming and very shrill.

An analysis of the specific loudness according to the DIN 4563 standard reveals systematic differences between the three identified main groups of fan sounds (A) unpleasant, (B) humming and (C) pleasant which might allow to derive new psychoacoustic indexes tailored to the characterization of fan noises.

COMPARISON OF SOUND QUALITY METRICS FOR AXIAL FLOW FANS WITH STRAIGHT AND FORWARD SWEPT BLADES

Jacques MUIYSER University of Stellenbosch (South Africa)
Johan VAN DER SPUY University of Stellenbosch (South Africa)
Anriëtte BEKKER University of Stellenbosch (South Africa)

The design of industrial axial flow fans is driven by regulations regarding the efficiency as well as the noise generated during operation. A number of aerodynamic design improvements exist that succeed in reducing fan noise. For example, it has been shown that forward swept fan blades exhibit reduced noise levels during operation when compared to a straight bladed fan. Even though the primary driver for reduced noise generation is the concern for hearing loss in humans, it has recently become important to consider customer perception of one's product in an increasingly competitive marketplace. As such, the purpose of this paper is to compare subjective sound quality metrics (loudness, sharpness, roughness, fluctuation strength and annoyance) of a straight bladed fan and one with swept blades.

Two 630 mm diameter fans with similar blade designs were tested in a ducted facility for a variety of flow rates where their performance as well as sound quality metrics could be determined. As expected, it was found that even though both fans performed similarly, the fan with swept blades exhibited a significantly lower level of tonal noise at the blade pass frequency as well as reduced broadband noise around 2 kHz when compared to the straight bladed fan. Upon investigation of the sound quality metrics it was found that, in addition to a reduction in loudness, the swept bladed fan also exhibited lower sharpness, roughness and fluctuation strength values for throttled flow rates, which are commonly associated with positive consumer impressions. However, with increased flow rates the difference between the two fans generally decreased, thereby indicating that the benefit of the improved sound quality exhibited by a swept bladed fan is dependent on its operating point.

SOUND REDUCTION BY LEADING EDGE SERRATIONS IN LOW-PRESSURE AXIAL FANS

Florian KRÖMER Friedrich-Alexander University Erlangen-Nürnberg (Germany)
Matthias WESTERMEIER Friedrich-Alexander University Erlangen-Nürnberg (Germany)
Andreas RENZ Friedrich-Alexander University Erlangen-Nürnberg (Germany)
Stefan BECKER Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

In recent years a trend towards modifications of the leading edge geometry in airfoils for reducing the sound emission and improving the stall behavior has been observed. However, low-pressure axial fan design methodologies for a decreased sound radiation almost rely exclusively on the use of fan blade skew rather than on leading edge modifications so far.

Hence, with this study we aimed to investigate the sound reduction capabilities of leading edge serrations applied to low-pressure axial fans in. For this, we designed a generic fan with interchangeable flat-plate fan blades. We then did a parametric study with focus on the aerodynamic and the acoustic fan properties with a reference fan blade set with straight leading edges and four fan blade sets that featured sinusoidal leading edges with varying serration wavelength and serration amplitude. The investigations were made under free inflow conditions and distorted inflow conditions with an increased inflow turbulence intensity, induced by a turbulence grid. On studying the inflow parameters, Laser Doppler anemometer measurements showed that the inflow turbulence intensity was increased by a factor of 2 compared with free inflow conditions, owing to the installation of the turbulence grid.

All fans with leading edge serrations showed better aerodynamic characteristics (pressure coefficient and efficiency) than the reference fan with straight leading edges along the aerodynamic characteristic curves. The fans with leading edge serrations additionally showed substantially lower overall sound pressure levels than the reference fan with straight leading edges over a broad operating range. The sound pressure spectra for both free and distorted inflow conditions revealed that tonal and broadband sound components are reduced by the leading edge serrations. Additionally, subharmonic tip noise that occurred for the reference fan was greatly suppressed by the leading edge serrations. We found that the serration wavelength was the main factor for the sound reduction, opposed to the serration amplitude for airfoils. Based on the parametric study, physical mechanism on the sound reduction by leading edge serrations are explained and recommendations on the optimal choice of leading edge serration parameters for decreasing the sound emission of low-pressure axial fans are given.

INFLUENCE OF THE ROTATION SPEED ONTO COMPACT AXIAL FANS BROADBAND NOISE AT CONSTANT DESIGN POINT

Julien GRILLIAT ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
Patrick BUCHWALD University of Stuttgart (ITSM) (Germany)
Damian VOGT University of Stuttgart (ITSM) (Germany)
Wolfgang LAUFER ebm-papst Sankt Georgen GmbH & Co, KG. (Germany)
Marc SCHNEIDER ebm-papst Mulfingen GmbH & Co. KG (Germany)
Andreas LUCIUS ebm-papst Mulfingen GmbH & Co. KG (Germany)
Michael SCHMITZ Siemens AG (Switzerland)

From an industrial point of view, fan size and particularly fan diameter is the major design constraint. Once this parameter is set, fan design consists in choosing the optimal couple between rotation speed and blade loading in order to achieve the target operating point. The influence of the rotation speed onto fan noise has been known for a long time. Moreover, recent studies clearly showed that blade loading affects fan broadband noise. The current knowledge can be resumed as follows: at too high speed, quadrupole related noise (turbulence, jet) will prevail and noise will rise. At too low speed, the blade loading will get high and the blade boundary layer very thick. Flow detachment or even stall is likely to happen, leading to excessive noise too. An optimum thus has to be found. Based on this conclusion, a two steps study has been carried out. The first step implied extensive CFD analysis and was presented in a previous conference. The second step aims at validating the numerical results and will be presented. A design point has been selected, and five geometries were developed for five different fan rotation speeds, from 7500 rpm to 10000 rpm.

The design rules as well as the CFD study will be summarized in the first part. Methods and major results will be presented. The numerical study did highlight an optimal rotation speed for the selected design rules. In a second part, the challenges met during the prototype realization phase will be discussed, and the technical choices made will be justified. Among others, CFD generated profiles often get very thin trailing edges, which are great challenges to manufacture. The experimental conditions will then be described. Finally, the experimental results will be presented and the comparison between CFD predictions and measurements will be discussed.

DESIGN OF A LOW SPEED RIM-SUPPORTED FAN FOR MINIMUM NOISE

Mark HURTADO Virginia Polytechnic Institute and State University (Virginia Tech) (United States)
Ricardo BURDISSO Virginia Polytechnic Institute and State University (Virginia Tech) (United States)

Axial flow fans are critical in maintaining a safe work environment by effectively circulating air in occupied areas. However, commercial ventilation fans are often loud and cause noise induced hearing loss from prolonged exposure. Consequently, there is an increasing need for quiet ventilation fans.

The need exists because many commercial ventilation fans are poorly designed aerodynamically and acoustically. Some of the major fan noise contributors are the rotor broadband self-noise, rotor-stator tonal interaction, tip gap noise, and blade steady loading and thickness. Rotor-stator and tip gap noise can be eliminated with a rim supported fan design. However, fan noise due to rotor self-noise, blade steady loading and thickness will not be eliminated for a rim supported fan. These noise sources are proportional to the 4-6th power of the fan tip speed, depending on the noise source. To this end, the present paper presents a control vortex design (CVD) methodology to design the fan blade profile to minimize fan tip speed and noise while preserving aerodynamic performance.

The CVD fan blades are characterized by a span wise changing circulation that ensures a higher flow rate contribution of the blade outer sections, i.e. axial flow increases from the blade hub to the tip. However, a non-uniform span wise circulation is susceptible to radially outward flow that increases near tip losses if the flow is not in radial equilibrium. Consequently, in this study the effect of radial flow is incorporated into the design procedure. To that end, the velocity profile was designed to maintain radial equilibrium and to maximizing the volumetric flow rate. The fan blade sections giving the desired velocity profile constitute the final fan blade design. The fan design has been integrated with an inlet section designed to maximize the fan aerodynamic performance. The fan and inlet section have been 3D printed. The fan is supported at the rim using an array of high speed miniature bearings.

Testing of the fan sound power level, mechanical power, flow rate and velocity profile have shown good agreement with predictions. It also shows better acoustic and aerodynamic performance than commercial fans. Consequently, a low speed rim supported CVD fan is a promising solution to the need of quiet ventilation fans.

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF THE VELOCITY FIELD IN FRICTION VENTILATORS

Andreas RENZ Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Julian PRASS Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Jörg RIEDEL Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Olaf NADLER Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)
Stefan BECKER Friedrich-Alexander-Universität Erlangen-Nürnberg (Germany)

In an effort to develop a new and simple concept for a decentralized ventilation system a cross flow friction ventilator was investigated. This friction ventilator consists of multiple circular discs which are driven by a motor and are rotating centrally in between two ducts (inlet and outlet duct of the ventilation system). The drag between the disc surfaces and the fluid induces a countercurrent flow in the two ducts while the discs also act as a heat exchanger between the two air flows. While the concept of the friction ventilator met the basic requirements for volume flow, pressure rise and heat recovery, the hydrodynamic efficiency found from experiments and simulations was lower than expected. In this study, we used Laser Doppler Anemometry to investigate the velocity field at different rotor geometries and operating points. Furthermore, we investigated the secondary currents in the ducts by means of numerical simulations in order to explain the low efficiencies.

The velocity field was measured in two areas, where on area of interest was the section between the discs themselves. The measurements there concentrated on the buildup of the wall boundary layer and its development along the disc. The other area was the velocity field up- and downstream of the friction ventilator.

As a result, we found that the wall boundary layer at the disc surface is thin compared to the distance between the discs and that a large part of the flow is only mildly affected by the rotating surfaces. The LDA measurements showed further that even in optimal operating points the flow field downstream of the friction ventilator was highly turbulent and could be non-uniform from disc duct to disc duct, depending on the rotor design. The numerical investigations showed that in almost unrestricted operation, secondary currents could be determined at velocities of up to 20 % of the mean main flow velocity, with secondary currents reaching up to 50 % in throttled operation. The investigations contributed to a better understanding of the velocity field of this kind of ventilator, also to the optimization of the rotor geometry and to the flow over rotating discs in general. Although the flow along the discs and the turbulent areas downstream were measured thoroughly, further investigations need to be made to fully understand the energy transfer from the disc to the air flow and the hydrodynamic losses that occur.

THE USE OF SERRATED LEADING EDGE FOR INFLOW CONDITIONING IN CENTRIFUGAL FAN

Lucio CARDILLO SED Soluzioni Srl (Italy)
Alessandro CORSINI Sapienza University of Rome (Italy)
Giovanni DELIBRA Sapienza University of Rome (Italy)
Anthony Geoffrey SHEARD AGS Consulting Llc (United States)
Lorenzo TIEGHI Sapienza University of Rome (Italy)

The design of air movement fans intended for industrial application is made challenging by the range of in-service conditions that must be accommodated. Each installation is unique, and typically constrained resulting in a distorted in-flow. The distorted in-flow classically results in lower fan efficiency. Severely constrained installations are characterised by in-flow dominated by separated flow features that degrade fan stability margin. Air movement fan designers must account for the impact of installation effects though both mechanically and aerodynamically robust design.

In this paper, we present a numerical study of a serrated centrifugal fan blade leading edge. The serrated feature minimizes the effect of distorted in-flow on fan stability and efficiency. The studied fan was designed for application in a roof-unit, laboratory tested with both original and serrated blade leading edges.

Numerical computations were performed using OpenFOAM with the cubic k-epsilon model of Lien and Leschziner. Performance of the studied fan was predicted with original and serrated blade leading edges with both un-distorted and distorted in-flow. Fan performance with two inlet distortions were predicted, with each distortion being chosen as characteristic of a different installation.

The paper concludes with a comparison of fan performance with and without the serrated blade leading edge when subjected to the un-distorted and two distorted in-flow. The impact of the serrated blade leading edge on fan stability and efficiency is considered, and the impact on in-service performance discussed. Implications for optimization of serrated blade leading edge geometry are then considered and the possible implication for long-term in-service reliability clarified.

FAN UNIT WITH SPECIAL GUIDE VANE DESIGN FOR LOW HUB RATIO

Frieder LÖRCHER Ziehl-Abegg SE (Germany)
Walter ANGELIS Ziehl-Abegg SE (Germany)

Low pressure axial fans for refrigeration, climatization and similar applications typically have low hub ratios of about 0.15..0.5. The reason for that is, on the one hand, that high volume flow rates are required at low pressure rise and, on the other hand, that the outer motor contour is used as impeller hub.

It is well known that fans with low hub ratio suffer from the phenomenon of “hub deadwater”, meaning the existence of a near-hub region of detached flow not contributing to the effective fluid transport from inlet to the outlet. This deadwater region is separated from the “sane” flow region by a separation layer within the fluid. The extent of this region grows with growing pressure rise required from the fan. The cross-section, which is effectively available for flow transport, is reduced by the detached flow region.

Classical guide vane design may leads to poor performances in such configurations. The reason for that is that by effect of guide-vanes, the cross-section of the hub deadwater may be increased reducing the effectively available cross section for the desired fluid transport. Consequently, at given volume flow rate, the axial flow velocity component increases. This effect can over-compensate the effect of reduction of the circumferential flow velocity component, resulting in flow acceleration and thus to a low or negative reaction ratio of the guide vanes.

In the present work, a stator fan unit including guide vanes and outer housing contour is designed which has primarily the effect to reduce the diameter of the hub detachment region. The cross-section effectively available for the fluid transport of the fan is additionally increased by using a diffusor contour of the housing. A part of the dynamic pressure available in the circumferential flow component is also reacted to static pressure rise.

The so designed fan unit leads to an important increase of static efficiency of the fan (20 %-30 %). The stall pressure point is significantly increased. Due to the reduced flow velocities, also the sound emissions are reduced significantly. The reduction of hub deadwater leads to more-compact, more cylindrical flow leaving the fan, reducing the risk of thermal short cuts in critical applications given by more divergent flow coming out of a fan without guide vane. We present in the paper comparisons of theoretical results obtained by simplified 1D-considerations, CFD results and experimental results.

STUDY ON NEW PARAMETERS FOR TONAL NOISE EVALUATION IN SMALL FAN NOISE

Gaku MINORIKAWA Hosei University (Japan)
Takefumi NAKANO Hosei University (Japan)

In the fields of sound quality evaluation in information technology (IT) equipment, parameters for tonal components such as TNR (Tone to Noise Ratio) and PR (Prominence Ratio) are standardized in ISO 7779, ECMA-74 and so on. TNR and PR are mainly focused to detect abnormal tonal noise in products for quality control. On the other hand, psychoacoustic parameter so called as Tonality was established by Terhardt and Aures. Tonality expresses a kind of tonal feeling for sound, however, it doesn't focus specific peak component.

The present study is an attempt to establish new parameters for tonal noise evaluation in small fan noise and other IT noise. The TTNR (Total Tone to Noise Ratio) and TPR (Total Prominence Ratio) are presented and calculation tool were developed. In the paper, relationships between subjective tonal feeling and these parameters calculated by various methodologies to pick up tonal components were examined. Sensory test was done by using a paired comparison method and method of limits.

Acoustic Perception of Irregularly Spaced Blades Fan

Xavier CARNIEL CETIM (France)
Alexis DOUCET CETIM (France)

For acousticians, noise reduction is not the unique issue considering fan noise. In applications where human beings are involved (work, domestic, medical), annoyance has to be considered when evaluating the impact of fan.

Blade passing frequency is well known to be a characteristic of fan noise. It is often considered as unpleasant because of its tonal characteristics. But what happens if blades are irregularly spaced?

Measurements have been performed on a test bench and on an earthmoving machine to compare noise levels and psychoacoustics criteria of fans with regularly or irregularly spaced blades. The paper presents the results obtained for these measurements.

TIME STRUCTURE ANALYSIS OF FAN SOUNDS

Carolin FELDMANN University of Siegen (Germany)
Joffrey LHONNEUR Ecole Centrale de Nantes (France)
Thomas CAROLUS University of Siegen (Germany)
Marc SCHNEIDER ebm-papst Mulfingen GmbH & Co. KG (Germany)

In general the sound of air delivering fans or fan systems is described by listeners utilizing descriptors like loudness, pitch, timbre, etc. but also by time-related adjectives such as rattling, fluttering or blowing. The time variance of a sound over time can lead to the perception of unpleasantness or annoyance, which in turn classifies a product as of minor sound quality. In this paper we investigate whether the concept of the psychoacoustic metric "roughness" is useful to evaluate time-related sound properties of fan sound with respect to its sensed quality. In a psychoacoustic analysis different algorithms for determining the metric "roughness" are in use, e.g. by Sottek, Daniel and Weber or Oetjen et al. They proved to be useful mainly for simple sinusoidal sounds. Modulation frequencies up to about 20 Hz are perceived as rhythm. The variation can be in frequency, e.g. used for a siren, or in amplitude, e.g. the audible beat when tuning a guitar. With increasing frequency the perception changes to impure or rough which is mathematically described by roughness. Typically, for complex sounds of fans the available algorithms are insufficient to yield values of roughness which reflect the results of jury tests.

In a first analysis, it was found that the sound of fan and fan systems is not modulated just by one modulation frequency. A huge amount of different solitary frequencies or narrow bands can be found, which also may vary randomly over time. For that, the statistical parameter "Shannon entropy", which is a metric for the randomness of distribution of events, was used. In a second step the perception of fan sounds with various different time structures is evaluated by jury tests. Finally, the difference between jury rating and the value of Sottek roughness is attributed to the different values of the Shannon entropy associated with each sound. In analogy to a study of Oetjen et al. we suggest and assess an improved concept of roughness for fan sound by taking into account the Shannon entropy, i.e. a parameter for the randomness of the modulation spectrum in time.

PERFORMANCE TESTING OF A RETROFITTED ACC FAN

Daniel ELS University of Stellenbosch (South Africa)
Jacques MUIYSER University of Stellenbosch (South Africa)
Johan VAN DER SPUY University of Stellenbosch (South Africa)
Chris MEYER University of Stellenbosch (South Africa)
François LOUW Kelvion Thermal Solutions (South Africa)
Albert ZAPKE ENEXIO Management GmbH (Germany)

The MinWaterCSP project aims to reduce the water consumption of concentrating solar power (CSP) plants by 75 to 95 % relative to wet cooling systems by introducing novel dry/wet cooling technology. This hybrid cooling system will make use of large axial flow fans to condense the process fluid. As part of the project a high efficiency axial flow fan was manufactured from a glass fibre reinforced polymer to demonstrate the possible improvements over a conventional air-cooled condenser (ACC) fan. The aerodynamic configuration of the so-called reference fan (R-fan) was that of a fan previously designed at Stellenbosch University that consisted of a larger hub to tip ratio than more prevalent ACC fans. Once manufactured the 30 ft R-fan was installed in the ACC of the Matimba power station for performance testing.

A single fan blade was instrumented with strain gauges to measure the blade loading in the flap- and chord-wise directions during operation. Strain gauges were also attached to the output shaft of the reduction gearbox to measure torque and anemometers were fitted to the outlets of the inclined heat exchanger bundles to measure air flow. The R-fan's performance was then measured over a period of three days where it was found that its stiff, lightweight blades experienced low dynamic blade loading with an amplitude of approximately 1 kN.m in the dominant flap-wise direction. This is due to the fact that the first natural frequency of the fan blades (4.6 Hz) is sufficiently far away from the second and third harmonics of the fan's operating speed. Furthermore, the mechanical power as measured on the output shaft of the gearbox was equal to approximately 145 kW for an average outlet air flow velocity of 3 m/s, indicating that the R-fan would be well suited as a full-scale ACC fan.

ENERGY SAVING AND ACOUSTICAL OPTIMIZATION: FAN RETROFIT FOR EXISTING INSTALLATIONS

Christian HAAG ebm-papst Mulfingen GmbH & Co. KG (Germany)

Estimations show that energy consumption of fans with electric drives will rise from 344TWh in 2005 up to 560TWh until 2020 within the EU. This corresponds to the energy production of roughly 140 large power plants (4 TWh per power plant). With improving the fans there is a chance for energy saving up to 34TWh until 2020. This corresponds to a reduction of the CO2-emission of 16 million tons per year or 8.5 power plants.

More than 50 % of the existing Climate-, Air Conditioning and Ventilation units in offices or industrial buildings are older than 25 years. During this period there was a noticeable increase in efficiency of modern fans related to optimizations of electric motors and improvements on the aerodynamic side.

The biggest potentials for energy saving lies in the replacement of antiquated fans and an adapted control-strategy that can deliver “tailored ventilation” based on the needs of the people who live or work inside the building. Replacing inefficient components is one way to improve existing units and typically much cheaper than buying a new unit. Therefore Retrofit of fans increases the lifetime of existing units for many years. As the replacement is done within the existing building environment usually no additional licenses are needed from public authorities for these projects.

A second way is to replace modules within the units that may have more functions integrated then just ventilation.

Retrofit itself will not only have a positive impact on environment and the financial expanses of companies, with the new and improved control function of the fans it will also increase the comfort for users and at the same time there is a chance to decrease noise level.

The presentation will show approaches to improve the energy efficiency and the acoustical behavior of fans in different applications. On the one hand there is room for improvement with regard to the electric drive (electric motor)

- Motor optimization: Replacement of AC-Motors (asynchronous Motor) with energy efficient EC-motors (synchronous motor). EC-motors use electronical commutation and permanent magnets to reduce losses within the motor
- Today’s EC-motors have different Control functions. Speed control can be used to decrease power consumption (Power consumption Pe ~ Speed n^3).

On the other hand aerodynamic optimization leads to higher overall efficiency and better noise values of fans

- Centrifugal fans: Change from inefficient forward-curved blowers to fans with backward-curved impellers. Fans with forward-curved impellers have high losses during conversion of dynamic pressure into static pressure. Fans with backward-curved impellers have much lower dynamic losses as static pressure is already generated within the impeller. With an additional scroll housing the dynamic energy of backward-curved fans can be further reduced to reach best efficiency values.
- Typical losses for axial fans are the kinetic energy at the outlet of the fan and the swirl created by the rotation of the fan. Increasing the outlet area (diffuser) will reduce the outlet air velocity, guide vanes can reduce the swirl. Both will reduce losses and therefore improve efficiency.

DESIGN OF NOISE REDUCED LARGE FANS FOR WIND TUNNEL APPLICATION WITH CFD-BASED OPTIMIZATION – A CASE STUDY

Konrad BAMBERGER University of Siegen (Germany)
Thomas CAROLUS University of Siegen (Germany)
Tilo KOZUSCHEK Howden Axial Fans GmbH (Germany)
Henrik TRYGGESON Howden Axial Fans AB (Sweden)

Aeroacoustic wind tunnels require not only appropriate passive sound attenuation measures but also low noise fans. Due to innovative manufacturing methods large fans may now be equipped with highly skewed rotor blades – a proven method for noise reduction. In this case study we describe the development steps of a complete axial fan stage comprising rotor and guide vanes at model scale level.

The reliable prediction of the acoustic emission of a fan is still computationally extremely ex-pensive. Therefore, to achieve the low noise target a substantial skew of the rotor blades and a combination of the counts of blade and guide vanes for minimizing mode propagation in the duct system are chosen. Naive blade skew can introduce 3D-flow effects with a substantial deg-radation of efficiency. Therefore, the rotor blade shape and the guide vanes are optimized aero-dynamically via an automated CFD-RANS-based optimization scheme.

The model scale fan stage is manufactured and its aerodynamic and acoustic characteristics are measured. A comparison with experimental data from a model scale state-of-the-art benchmark wind tunnel fan with unskewed rotor blades reveals that the new optimized design has a consid-erably larger range of operation with good efficiency and without stall. The sound power emit-ted by the optimized fan is substantially lower as from the benchmark over the complete range of possible operation.

NOISE REDUCTION OF A LARGE AXIAL FLOW FAN FOR CSP AIR-COOLED CONDENSERS

Gino ANGELINI Sapienza University of Rome (Italy)
David VOLPONI Sapienza University of Rome (Italy)
Mike WILKINSON Stellenbosch University (South Africa)
Sybrand VAN DER SPUY Stellenbosch University (South Africa)
Tommaso BONANNI Sapienza University of Rome (Italy)
Lorenzo TIEGHI Sapienza University of Rome (Italy)
Giovanni DELIBRA Sapienza University of Rome (Italy)
Alessandro CORSINI Sapienza University of Rome (Italy)
Theodor VON BACKSTRÖM Stellenbosch University (South Africa)

The MinWaterCSP project aims to reduce the water consumption of concentrating solar power (CSP) plants by 75 to 95 % relative to wet cooling systems and to improve plant efficiency by 2 to 3 % relative to current dry-cooled systems by introducing novel dry/wet cooling technology. This hybrid cooling system will make use of large axial flow fans to condense the process fluid. In particular, these fans will be used in large arrays (with hundreds of devices) so it is important to increase the efficiency of the fans, but also to reduce as much as possible their acoustic emissions.

As part of the MinWaterCSP project, Stellenbosch and Sapienza Universities are collaborating to design a high-efficiency low-noise fan for air-cooled condensers.

In this paper we focus on two different strategies to reduce the noise of the M-Fan, i.e. the final prototype produced by the aerodynamic design process for MinWaterCSP. The first round of optimization was carried out coupling an axis-symmetric flow solver (AxLab) with an optimization software based on brute force optimization. The procedure aimed at optimizing the chord and pitch distribution of the M-Fan to reduce trailing edge noise and increase the fan efficiency. The component of noise related to the wake shed by the trailing edge was computed according to the model of Fukano et al. [1]. Flow field quantities necessary to the calculations were computed by AxLab. The optimization methodology was based on a brute-force algorithm that allowed chord and pitch values of xxx control sections to vary to xxxx discrete values. The total number of tested individuals was limited by imposing manufacturability contraints to the pitch and chord distributions. The second approach was based on challenging the paradigm of design vortex distribution of work along the blade span. In particular a power law distribution was selected, aiming at unloading the tip and control the tip leakage vortex.

TURBO-MACHINERY DESIGN BASED ON MULTI-PHYSICS FLUID-STRUCTURE OPTIMIZATION

Kossivi GOKPI INTES FRANCE (France)
Jacques MARCHESINI INTES FRANCE (France)
Bruno DEMORY Valeo Thermal Systems (France)
Manuel HENNER Valeo Thermal Systems (France)

Improving efficiency of fan systems is of great interest in the automotive industry. Turbomachine optimization is becoming a key technique to improve fan systems guarantying good performances in terms of aerodynamics, structure and vibration as well as the reduction of noise.

This paper aims to show the geometric parameterization and mesh morphing capability of a large number of parameters for the design and optimization of turbomachine. It could apply for a wide range of cases as demonstrated by realistic examples from industry. It also shows the use of multi-physics approach such as fluid-structure interaction in the design and optimization of fans. Up to fifteen parameters such as chords, stagger and sweep angles, camber, profile thickness have been used on a single blade to control the blade geometry. These independent parameters are set on an initial mesh, and a morphing technique drives the blade modification regarding the parameter combination. This allows designing various types of complex blade shapes with smoothness mesh quality verification during the morphing process. The results are discussed with particular attention to the static deformations under centrifugal effect, the eigenmodes as well as the displacements generated. The shape deformations induced by these parameters variations is transposed to the Computational Fluid Dynamic (CFD) mesh. Steady and unsteady RANS simulations are performed to predict the aerodynamic performances and to extract the pressure field variation in time on the fan. A structural analysis can then be conducted in order to calculate the centrifugal forces and the harmonics for the structural dynamic response of the rotor. The structural noise and its sound radiation is then evaluated.

Results obtained have allowed evaluating the set of parameters that influence the structural noise on the blade. The optimization process is then conducted by performing an automatic differentiation of the selected relevant parameters which enables to build database containing high order values of these parameters for decision support.

DEVELOPMENT, VALIDATION AND APPLICATION OF AN OPTIMIZATION SCHEME FOR IMPELLERS OF CENTRIFUGAL FANS USING CFD-TRAINED METAMODELS

Konrad BAMBERGER University of Siegen (Germany)
Julian BELZ University of Siegen (Germany)
Thomas CAROLUS University of Siegen (Germany)
Oliver NELLES University of Siegen (Germany)

Cost-effective optimization of centrifugal impellers requires quick and reliable methods to evaluate the objective function. A typical objective function is maximization of efficiency for a given design point. In this work, we suggest to use CFD-trained metamodels which evaluate the objective function several orders of magnitude faster than CFD itself. The metamodels used are artificial neural networks (ANN) and differ from previously developed metamodels in terms of universality since they can be used for optimizing all typical design points of centrifugal impellers according to the Cordier diagram. In addition, the optimization scheme is supposed to handle typical operational and constructive constraints.

The biggest challenge in the development of the metamodels is to ensure adequate accuracy with limited computational resources required to generate the CFD dataset used to train the metamodels. For that purpose, the geometry was parameterized such that a large geometrical diversity (and hence the realization of very different design points) can be achieved with only nine independent geometrical parameters. The number of required geometry variations was reduced by a two-stage Design of Experiment consisting of a passive and an active learning phase. In the passive phase, the geometrical parameters were varied in a space-filling manner, i.e. only the inputs of the metamodels were considered. In the subsequent active phase, the aerodynamic performance (i.e. the metamodel outputs) was considered, too. The main target of the active phase was to focus mostly on those areas of the input space in which the (preliminary) metamodels have the lowest quality and on areas of high impeller efficiency since these areas are most relevant for the purpose of aerodynamic optimization. The computational time for the CFD simulations was kept to a minimum by using the comparatively cheap RANS method and by optimizing the computational grid with the aim of matching experimental performance data of three prototypes with as few grid points as possible.

The new optimization scheme was applied to numerous design points and the resulting geometries were simulated by CFD. It was found that the metamodels overpredict efficiency at design points with untypically small pressure coefficients. At standard design points and untypically large pressure coefficients, however, CFD confirms well the metamodel predictions proving the broad applicability of the new optimization scheme. In addition, four optimal geometries were manufactured and examined experimentally to validate the CFD method and to prove that the optimization scheme does not exploit weak points of the CFD model.

SPONSORS

&

EXHIBITORS

SPONSORS

Conference Gala Diner Sponsor

ebm papst
Bachmühle 2
74673 Mulfingen, Germany

Website: www.ebmpapst.com

Badges Sponsor

Ziehl-Abegg
Heinz-Ziehl-Straße
74653 Künzelsau, Germany

Website: www.ziehl-abegg.com

Proceedings and Book of Abstracts Sponsor

Boldrocchi S.r.l
Viale Trento e Trieste 93
20853 Biassono (MB), Italy

Website: www.boldrocchi.eu/

Welcome Drinks Sponsor

Nicotra Gebhardt S.p.A
Via Montenapoleone, 9
20121 Milano, Italy

Website: www.nicotra-gebhardt.com

Lanyards Sponsor

AMCA
30 West University Drive
Arlington Heights,
IL 60004, USA

Website: www.amca.org

Delegate Bags sponsor

Avanced Design Technology
Dilke House, 1 Malet Street
London, UK
WC1E 7JN

Website: www.adtechnology.co.uk

Insert in Delegate Bags

CFturbo Software & Engineering GmbH
Unterer Kreuzweg 1
D-01097 Dresden, Germany

Website: www.cfturbo.com

EXHIBITORS

Advanced Design Technology

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

AGS Consulting, LLC
P.O. Box 79267
Atlanta, GA 30357
Phone: +1-678-974-9565
E-mail: geoff@agsconsultingllc.com
Website: www.agsconsultingllc.com

AMCA

30 West University Drive
Arlington Heights
IL 60004
USA
Mark Stevens, Deputy Executive Director
Phone: +01 847 704 3290
E-mail: mstevens@amca.org
Website: www.amca.org

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

Centre Technique des Industries Mécaniques
Pôle "Mesures-Mécatronique Acoustique Vibrations"
52 Avenue Félix Louat, BP 80067
60304 Senlis
France
Céline Cammarata
Phone: +33 344 67 35 11
E-mail: celine.cammarata@cetim.fr
Website: www.cetim.fr

EXHIBITORS

CFturbo Software & Engineering GmbH

Headquarters
Unterer Kreuzweg 1
D-01097 Dresden
Germany
Phone: +49 351 40 79 04-79
E-mail: sales@cfturbo.de
Website: www.cfturbo.com

Datatel - Telemetrie Elektronik GmbH

Berliner Allee 42
D-30855 Langenhagen
Germany
Phone: +49 (0) 511 / 97 83 96-0
E-mail: info@datatel-telemetry.de
Website: http://www.datatel-telemetry.de/en/

EP-E: Ehrler Prüftechnik Engineering GmbH

Wilhelm-HachPhone-Str. 8
97996 Niederstetten
Germany
Oliver Hammel, CEO, Managing Director
Phone: +49 (0) 7932 / 60 666 40
Website: www.ep-e.com

Exa Corporation

55 Network Drive, Burlington, MA USA

Exa GmbH, Landshuter Allee 8,
80637 München, Germany
Phone: +49 (0) 89 1895073-20
Website: www.exa.com

HEAD acoustics GmbH

Ebertstraße 30a
52134 Herzogenrath
Germany
Phone: +49 2407 577-0
E-mail: info@head-acoustics.de
Website: www.head-acoustics.de

Industrial Science GmbH

Alexanderstraße 25
D-64283 Darmstadt
Germany
Contact: Mehdi Nakhjiri
Phone: +49 6151 787-7000
E-mail: info@industrial-science.de
Website: www.industrial-science.de

EXHIBITORS

MSC Software FFT

Axis Park Louvain-la-Neuve
9 rue Emile Francqui
B-1435 Mont-Saint-Guibert
Belgium
Phone: +32 10 45 12 26
E-mail: contact@fft.beValerie Larsille
Website: www.fft.be

Nicotra Gebhardt S.p.A.

Via Montenapoleone, 9
20121 Milano
Italy
Contact: Achim Falk
Phone: +49 (0)7942 101 100
a.falk@nicotra-gebhardt.com
Website: www.nicotra-gebhardt.com

Numeca

Chaussée de la Hulpe, 189
Terhulpsesteenweg
B-1170 Brussels
BELGIUM
Contact: Gabriele Kramer
Phone: +49 9187 410 310
Phone: +32.2.647.83.11
E-mail: sales@numeca.be
Website: www.numeca.com

OROS GmbH

Hofstraße 272
D-56077 Koblenz
Germany
Contact: Günther Ring
E-mail: info@oros-deutschland.com
Phone: +49 (0)261 1 33 96 50
Website: www.oros-deutschland.com

Polytec GmbH

Polytecpl. 1-7,
76337 Waldbronn
Germany
E-mail: info@polytec.de
Phone: +49 (0) 7243 604-0
Website: www.polytec.com/de