fan2012

International Conference on Fan Noise, Technology and Numerical Methods

Abstract Book

18-20 April 2012
Senlis, France

www.fan2012.org

images copyright of Office Tourisme Senlis

cetim

CETIAT
ensemble, innover et valider

Institution of MECHANICAL ENGINEERS

CD PROCEEDINGS AND ABSTRACT BOOK SPONSOR

FläktWoods

ISBN Number: 978-0-9572374-0-7

The Abstract Book and CD Proceedings are sponsored by:

Fläkt Woods provides air movement products and services including fans for cooling systems, and also for increasing the efficiency, environmental performance and safe operation of equipment and processes. We have solutions for every industrial application, such as Power Generation, Tunnel, Metro, Cement, Steel, Mining, Traction and Transformers.

Fläkt Woods also offers a wide range of axial flow fans for general air movement and special applications such as emergency high temperature smoke extract and pressurisation. ATEX fans for explosive atmospheres and multistage options for increased pressure development.

www.flaktwoods.com

CONTENTS

INTRODUCTION FROM THE CONFERENCE CHAIR

Welcome to Fan 2012. The Fan 2012 conference provides all those who attend with a unique opportunity to network with others working in the fan industry. This three-day conference includes keynote lectures, technical presentations organised as three parallel sessions, focusing on fan technology, fan noise and theoretical methods. Panel sessions will also provide you with an opportunity to learn more about the forthcoming Energy using Product (EuP) Directive and how computational methods are impacting the fan design process.

The conference exhibition provides you with access to those organisations 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 2012.

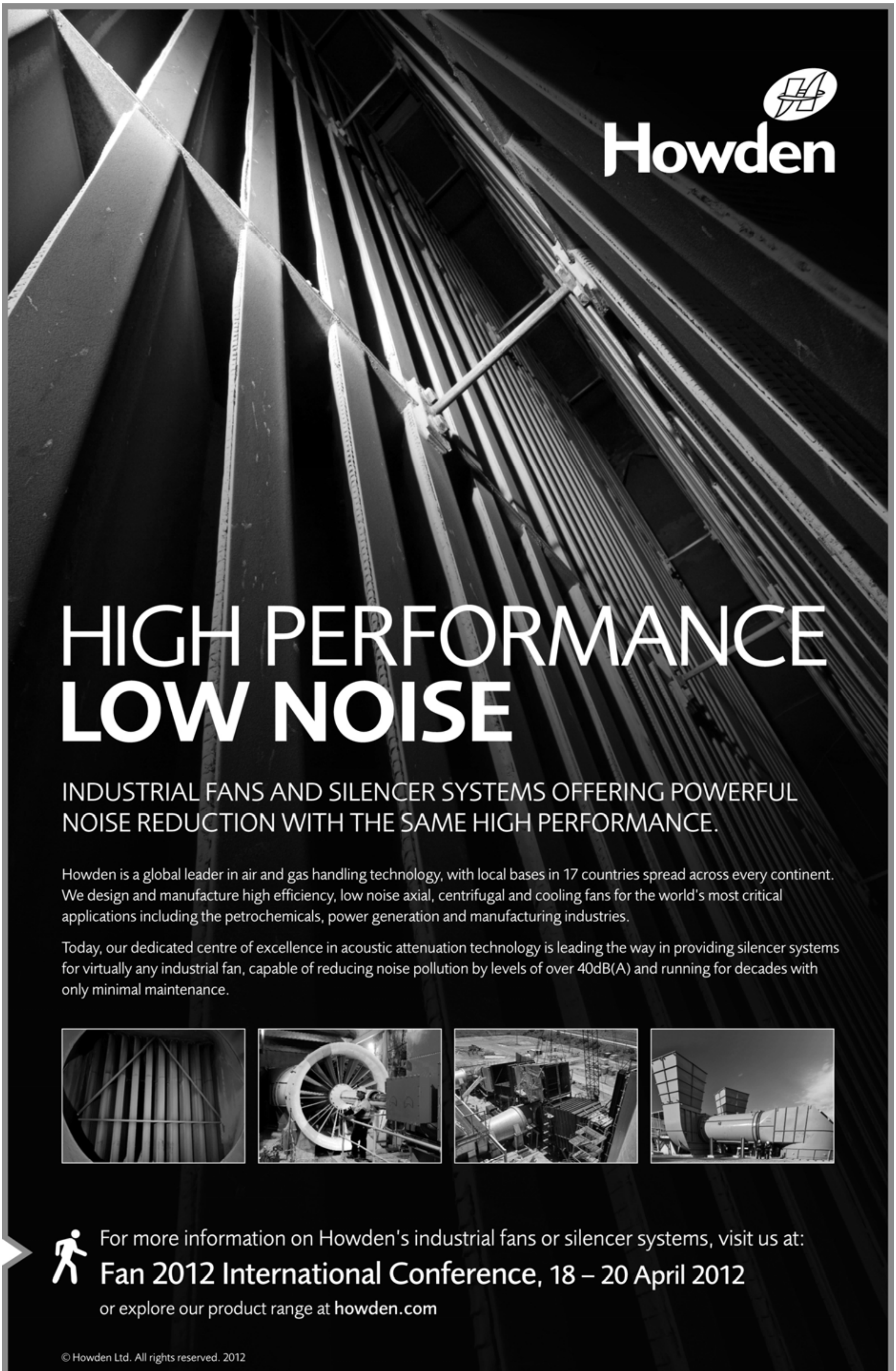
The location of Fan 2012 conference, Senlis is a French commune in the Oise Department in the Picardie Region near Paris. It has a long and rich heritage. The medieval town is where the monarchs of the early French dynasties lived, attracted by the proximity of the Chantilly forest. I am sure that you will enjoy not only the conference itself, but also the location.

The Fan 2012 conference is important because the design of fans is constantly evolving to meet the ever-increasing demands for higher efficiency machines, combined with the requirements for lower noise and high availability. In addition many fans are now being used in safety related applications, such as smoke control in buildings and underground spaces in the event of a fire. Variable speed is now more common, leading to some additional design problems and many existing users are revisiting their plant looking at ways to upgrade the fans. The use of numerical simulation techniques is has also become an established part of aerodynamic design processes.

This conference is a forum for fan and system designers, manufacturers and operators. It will improve all our understanding of fans and their system interaction, and in so doing is not just good for those who attend the conference, but for the wider fan industry. I passionately believe that "a high tide lifts all boats". When we come to look back on the Fan 2012 conference I am more than optimistic that it will be an event all those who attended regard as a high-point not just in 2012, but in the in the history of the fan industry.

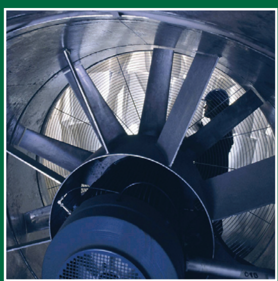
I would like to take those opportunity for and on behalf of the Fan 2012 organising committee to welcome you to Senlis, and to thank you for attending Fan 2012. Take your time to attend the sessions, listen to what is said and challenge what you hear. Above all, renew old acquaintances, form new friendships and enjoy the conference.

Geoff Sheard
Chairman – Fan 2012 Organising Committee


Howden
HIGH PERFORMANCE
LOW NOISE
INDUSTRIAL FANS AND SILENCER SYSTEMS OFFERING POWERFUL NOISE REDUCTION WITH THE SAME HIGH PERFORMANCE.
Howden is a global leader in air and gas handling technology, with local bases in 17 countries spread across every continent. We design and manufacture high efficiency, low noise axial, centrifugal and cooling fans for the world's most critical applications including the petrochemicals, power generation and manufacturing industries.
Today, our dedicated centre of excellence in acoustic attenuation technology is leading the way in providing silencer systems for virtually any industrial fan, capable of reducing noise pollution by levels of over 40dB(A) and running for decades with only minimal maintenance.
For more information on Howden's industrial fans or silencer systems, visit us at:
Fan 2012 International Conference, 18 – 20 April 2012
or explore our product range at howden.com
© Howden Ltd. All rights reserved. 2012

World Class Tunnel Ventilation Solutions

by Fläkt Woods Limited



Large Aerofoil Axial Flow Fans

- Sizes: up to Ø 4000mm
- Volume flow: up to 425m³/s
- Total pressure: up to 3400 Pa
- High temperature capability:
  - – up to 250°C for 2 hours
  - – options up to 400°C for 2 hours
- Wide range of fans already certified EN 12101-3 and ISO 21927-3 by a third party
- Role: Heavy duty industrial and infrastructure applications
  - – higher pressure/volume
- Option:Fully-reversible flow

Contact us to find out further information

www.flaktwoods.com

Fläkt Woods Limited
Axial Way, Colchester, Essex, CO4 5ZD, United Kingdom

Tel: + 44 (0)1206 222 555 Fax: + 44 (0)1206 222 777
email: marketing.uk@flaktwoods.com web: www.flaktwoods.com

FläktWoods

ORGANISING COMMITTEE

Geoff Sheard (Fläkt Woods, UK) – Conference Chairman
Alessandro Corsini (Sapienza University of Rome, Italy) – Theoretical Methods & CFD Track Chairman
Alain Guédel (CETIAT, France) – Fan Noise Track Chairman
Ron Mulholland (Howden, UK) – Fan Technology Track Chairman
François Bessac (CETIAT, France)
Amelia Brunt (IMechE, UK)
Xavier Carniel (Cetim, France)

SCIENTIFIC ADVISORY COMMITTEE

Mats Åbom (KTH – The Marcus Wallenberg Laboratory, Sweden)
Didier Amor (Fläkt Woods, France)
Roberto Arias (Zitron, Spain)
Guido Banzi (Maico Italia, Italy)
Mike Barry (Twin City, USA)
Stefano Bianchi (Sapienza University of Rome, Italy)
Domenico Borello (Sapienza University of Rome, Italy)
Tony Breen (Nuaire, UK)
Dario Brivio (Nicotra, Italy)
Klaus Brun (IGTI Board of Directors, USA)
Guy Caignaert (ENSAM, France)
Thomas Carolus (University of Siegen, Germany)
Jan Cermak (ACME Fan, USA)
Bill Cory (Consultant, UK)
Alice Debonnet-Lambert (CIDB, France)
Marius Gamissans (Soler & Palau, Germany)
Alain Godichon (FläktWoods, France)
Yvon Goth (Cetim, France)
Mohamed Hamdi (University of Compiègne, France)
Lixi Huang (Hong Kong University, China)
Johannes Hyrynen (VTT, Finland)
Iain Kinghorn (FläktWoods, UK)
Chris Lawn (Journal of Power & Energy, UK)
Geoff Lockwood (ebm-papst, UK)
Alan Macklin (ELTA Group, UK)
Costantino Mogentale (Ferrari Ventilatori Industriali, Italy)
Young Moon (Korea University, Korea)
Stéphane Moreau (University of Sherbrooke, Canada)
Said Naji (Valéo, France)
Ralf Neumeier (Ziehl-Abegg, Germany)
Franco Rispoli (Sapienza University of Rome, Italy)
Michel Roger (Ecole Centrale de Lyon, France)
Omar Sadi (Ziehl-Abegg, Germany)
Mats Sandor (System Air, Sweden)
Jürgen Schöne (ebm-papst, Germany)
Colin Scrivener (ETC Organising Committee, UK)
Mark Stevens (AMCA International, USA)
Jean Tourret (INCE Europe, France)
János Vad (Budapest University, Hungary)
Paul Wenden (Fläkt Woods, UK)
Henrik Witt (Witt & Sohn, Germany)

Introducing CETIAT

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

CETIAT has an extensive and practical knowledge of the aerodynamic and acoustic performance of fans, especially axial and centrifugal fans. In this field, as in others, CETIAT carries out research programs (technical know-how compilation, design tools development, expertise capitalisation and development of fan sound measurement standards,…). These acquired skills 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 others fields of activity where CETIAT skill and expertise may be useful, such as in the automotive industry for instance.

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

- Staff training;
- Technological survey;
- Design optimisation and integration of fans using 3D flow and heat transfer models and several metrological tools (clinometer, laser Doppler anemometry, acoustic and vibration sensors, …);
- Aerodynamic and acoustic fan performance measurement in accordance with ISO 5801, ISO 13347 and ISO 5136;
- Measuring instruments calibration.

Introducing 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.

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:

Patrick Vincent
Tél. : 33(0)344673682 sqr@cetim.fr

Institution of MECHANICAL ENGINEERS

Introducing The Institution of Mechanical Engineers

The Institution of Mechanical Engineers is the fastest growing professional engineering institution in the UK. Our 100,000 members work at the heart of the country's most important and dynamic industries.

With a 160-year heritage supporting us, today's Institution is a forward-looking, campaigning organisation. By working with leading companies, universities and think tanks, we create and share knowledge to provide government, businesses and the public with fresh thinking and authoritative guidance on all aspects of mechanical engineering.

We truly believe we can improve the world through engineering. So the Institution finds and nurtures new talent, helping engineers build their careers and take on the challenges that, when solved, will make a difference to all of us.

In the UK, engineering has achieved great successes, but in a quiet way. We're looking to shout about the achievements of our members and the industry, taking a positive, inspiring message into schools and out into the media. By being independent of both government and business, and avoiding strategic relationships with single-issue bodies or pressure groups, we can deliver genuinely impartial advice in a passionately committed manner.

As an Institution, we focus on four principal themes which affect and are affected by our engineers:

- Energy
- Environment
- Transport
- Education

Our Divisions and Groups however feature on numerous areas of mechanical engineering: www.imeche.org/knowledge/industries

The Institution of Mechanical Engineers
1 Birdcage Walk
Westminster
London, UK
SW1H 9JJ
www.imeche.org

Tel: +44 (0)20 7222 7899
Email: enquiries@imeche.org

SUPPORTING ORGANISATIONS

AFM: Association Française de Mecanique

AMCA International: Air Movement and Control Association International

ASHRAE: American Society of Heating, Refrigerating and Air-Conditioning Engineers

CIDB: Centre d'Information et de Documentation sur le Bruit

EUROVENT: European Committee of Air Handling and refrigeration Equipment Manufacturers

FETA: Federation of Environmental Trade Associations

Fan Manufacturers Association

FMA ANZ: Fan Manufacturers Association of Australia and New Zealand

HEVAC: Heating, Ventilation and Air-Conditioning Manufacturers Association

IGTI: International Gas Turbine Institute

INCE Europe: Institute of Noise Control Engineering – Europe

SFA: French Acoustics Society

Uniclima

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

LIST OF SESSIONS

00 - Keynote Lectures

Noise Proves Nothing - Sources of Fan Noise and Their Prediction

- CAROLUS Thomas – University of Siegen (Germany)

Inlet Flow distortion and Lip Separation in Ducted Fans used in VTOL UAV Systems

- CAMCI Cengiz – Penn State University (USA) (United States)
- AKTURK Ali – Siemens Energy (United States)

EU Policies on Energy Efficiency and the Impact on Fan Manufacturers

- DE WILT Guido – Policy Officer - Energy Efficiency - European Commission (Directorate - General for Energy) (Belgium)

On the role of CFD in the aerodynamic development of industrial fans or from equations to colors (and back again)

- CORSINI Alessandro – Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)

18 April 2012

A1 Tonal Noise Control with Flow Obstructions

Fan Tonal Noise Reduction using Calibrated Obstructions in the Flow - An Experimental Approach

- GOTH Yvon – CETIM (France)
- BESOMBES Michel – CETIM (France)
- CHASSAIGNON Christian – DYVA (France)
- GERARD Anthony – Université de Sherbrooke (Canada)

Numerical Optimization of the Tonal Noise of a Backward Centrifugal Fan Using a Flow Obstruction

- PEROT Franck – Exa Corporation (United States)
- KIM Min-Suk – Exa Corporation (United States)
- LE GOFF Vincent – EUROXA (France)
- CARNIEL Xavier – CETIM (France)
- GOTH Yvon – CETIM (France)
- CHASSAIGNON Christian – DYVA (France)

Tonal Noise Control from Centrifugal Fans Using Flow Control Obstructions

- GERARD Anthony – Université de Sherbrooke (Canada)
- BESOMBES Michel – CETIM (France)
- BERRY Alain – Université de Sherbrooke (Canada)
- MASSON Patrice – Université de Sherbrooke (Canada)
- MOREAU Stephane – Université de Sherbrooke (Canada)

A2 Fan Mechanical Design

Innovative Fan Drives for Hybrid Trucks

- JUSTIN Thomas – Volvo 3P (France)
- MORGAN Tessa – Volvo 3P (France)
- BRALY Julien – Volvo 3P (France)

Designing a Centrifugal Fan of Carbon Fiber Laminate for High Circumferential Speed

- RATTER Heiko – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
- MAUCH Herrmann – Hürner-Funken GmbH (Germany)
- HANGS Benjamin – Fraunhofer-Institude for Chemical Technology (Germany)
- ÇAĞLAR Şaban – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
- GABI Martin – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)

Use of Grade 80 Steel in Fabricated Centrifugal Impeller

- BRUNCK Nicholas – American Fan Company (United States)
- OKELEY Paul – Fläkt Fan Group (United States)
- SHEARD Geoff – Fläktwood Ltd (United Kingdom)

A3 Numerical Optimisation of Centrifugal Fan Performances

Maximum Achievable Efficiency of Centrifugal Fans Without Housing

- XIA Yingan – Punker GmbH (Germany)
- ÇAĞLAR Şaban – Karlsruhe Institut für Technologie (Germany)
- RATTER Heiko – Karlsruhe Institut für Technologie (Germany)
- GABI Martin – Karlsruhe Intitute für Technologie (Germany)

Design Improvements of Sirocco Type Fans by Means of Computational Fluid Dynamics and Stereo Particle Image Velocimetry

- FRANK Stefan – University of Applied Sciences HTW Berlin (Germany)
- DARVISH Manoochehr – University of Applied Sciences HTW Berlin (Germany)
- TIETJEN Bastian – University of Applied Sciences HTW Berlin (Germany)

Fluid Dynamic Design and Optimization of a Double Entry Fan Driven by Tractor Power Take Off for Mist Sprayer Application

- PINELLI Michele – University of Ferrara (Italy)
- MORINI Mirko – University of Ferrara (Italy)
- ROSSINI Mauro – Ideal Srl (Italy)
- FERRARI Cristian – University of Ferrara (Italy)

B1 Computational Aeroacoustics

Computational Aeroacoustic Analysis of an Industrial Fan Application using a Hybrid Approach

- CABROL Marie – Free Field Technologies (Belgium)
- LENEVEU Romain – Free Field Technologies (Belgium)
- D'UDEKEM Diego – Free Field Technologies (Belgium)
- MANERA Julien – Free Field Technologies (Belgium)
- DETANDT Yves – Free Field Technologies (Belgium)
- MENDONCA Fred – CD-Adapco (United Kingdom)

Experiments and Numerical Predictions of Flow Rates and Aero-Acoustics from Small Radial Notebook Blowers

- GULLBRAND Jessica – Intel Corporation (United States)
- BELTMAN Willem – Intel Corporation (United States)

Optimization of Fan Noise by Coupling 3D Inverse Design and Automatic Optimizer

- ZANGENEH Mehrdad – University College London (United Kingdom)
- DE MAILLARD Matthieu – Advanced Design Technology Ltd (United Kingdom)

B2 HVAC and Condenser Fans

Fan-System Interaction and Blockage Effects for HVAC Multi-Fan Units

- VELARDE-SUAREZ Sandra – Universidad de Oviedo (Spain)
- GUERRAS-COLON F. Israel – Universidad de Oviedo (Spain)
- GONZALEZ Jose – Universidad de Oviedo (Spain)
- ARGÜELLES DIAZ Katia M. – Universidad de Oviedo (Spain)
- FERNANDEZ ORO Jesus M. – Universidad de Oviedo (Spain)
- SANTOLARIA MORROS Carlos – Universidad de Oviedo (Spain)

Simultaneous Measurement of Air Flow and Blade Loading Conditions in an Air-Cooled Steam Condenser Fan

- MUIYSER Jacques – University of Stellenbosch (South Africa)
- ELS Danie N. J. – University of Stellenbosch (South Africa)
- VAN DER SPUY Sybrand J. – University of Stellenbosch (South Africa)

Testing an Axial Flow Fan Designed for Air-Cooled Steam Condenser Application

- VAN DER SPUY Sybrand J. – Stellenbosch University (South Africa)
- VON BACKSTRÖM Theodore W. – Stellenbosch University (South Africa)
- KRÖGER Detlev G. – Stellenbosch University (South Africa)
- BRUNEAU Phillipe R.P. – Sasol (South Africa)

Simplified Theory of a Risk of Thermal Catastrophe in Air-Cooled Electronic Circuits

- VINOKUR Roman – Resmed Motor Technologies (United States)

B3 Unsteady Aerodynamics and Flow Simulations

On the Role of Leading-Edge Bumps in the Control of Stall On-Set in Axial Fan Blades

- CORSINI Alessandro – Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)
- DELIBRA Giovanni – Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)
- SHEARD Geoff – Fläkt Woods Limited (United Kingdom)

3D Unsteady CFD Simulation of the Unsteady Flow in a Centrifugal Fan

- YANG Yang – Clausthal University of Technology (Germany)
- LUCIUS Andreas – Clausthal University of Technology (Germany)
- BRENNER Gunther – Clausthal University of Technology (Germany)

Investigation of Unsteady Flows in a Centrifugal Fan using High-Speed PIV and Numerical Simulations

- LUCIUS Andreas – Clausthal University of Technology (Germany)
- BRENNER Gunther – Clausthal University of Technology (Germany)
- LEHWALD Andreas – Otto-von-Guericke University (Germany)
- THEVENIN Dominique – Otto-von-Guericke University (Germany)

Investigation on the Unsteady Aerodynamics of an Industiral Fan

- BORELLO Domenico – Sapienza University (Italy)
- CORSINI Alessandro – Sapienza University (Italy)
- RISPOLI Franco – Sapienza University (Italy)
- SHEARD Geoff – Fläkt Woods Ltd (United Kingdom)

C1a Tonal Noise Reduction

Tonal Noise of an Isolated Axial Fan Rotor due to Inhomogeneous Coherent Flow Structures at the Intake

- STURM Michael – University of Siegen (Germany)
- CAROLUS Thomas – University of Siegen (Germany)

An Investigation of the Effect of Uneven Blade Spacing on the Tonal Noise Generated by a Mixed Flow Fan

- DESVARD Ludovic – Dyson Ltd (United Kingdom)
- HURAULT Jeremy – Dyson Ltd (United Kingdom)
- MOHAMED ZAMZAM Muhammad Affendi Bin – Dyson Ltd (United Kingdom)
- SYMES Ash – Dyson Ltd (United Kingdom)

C1b Noise Control by Passive Means

Investigation into the Sound Power Level Reduction Achieved using Acoustic Jackets on a Centrifugal Fan and an Axial Fan

- SIMMS Mark – Ventac Group (Ireland)

Fan Noise Control: Case of Data Centres

- VERLHAC Jean-Marie – CETIM (France)

Suppression of Fan Noise of High Circumferential Mode Using Optimized Duct Wall Properties

- HUANG Lixi – Department of Mechanical Engineering, The University of Hong Kong (China)

C2 Performance Measurements

The Influence of the Reynolds Number and Roughness on the Efficiency of Axial and Centrifugal Fans – A Physically Based Scaling Method

- PELZ Peter – Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)
- STONJEK Stefan – Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)
- MATYSCHOK Berthold – Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)

Experiments on the Use of Symmetrized Dot Patterns for In-Service Stall Detection in Industrial Fans

- BIANCHI Stefano – Sapienza University (Italy)
- SHEARD Geoff – Fläkt Woods (United Kingdom)
- CORSINI Alessandro – Sapienza University (Italy)

Inlet Installation Effects on Different Types of Fans and Ductwork Designs

- GUEDEL Alain – CETIAT (France)
- CORY Bill – William Cory Consultancy (United Kingdom)
- STEVENS Mark – AMCA International (United States)

C3 CFD Validation Studies

CFD Electric Motor External Fan System Comparison and Validation

- CEZARIO Cassiano A. – WEG Equipamentos Eletricos - Motores (Brazil)

Numerical Investigation of Axial Fans in Serial Connection

- AXTMANN Gabriel – University od Stuttgart (Germany)
- SCHMITZ Michael – ebm-papst St. Georgen GmbH & Co. KG (Germany)
- LAUFER Wolfgang – ebm-papst St. Georgen GmbH & Co. KG (Germany)

CFD Techniques Applied to Axial Fans Design of Electric Motors

- BORGES Samuel Santos – Weg Equipamentos Eletricos - Motores (Brazil)

Numerical and Experimental Investigations of Single-Flow and Bypass-Flow Fans

- MILESHIN Victor – Central Institute of Aviation Motors (CIAM) (Russia)
- OREKHOV Igor – Central Institute of Aviation Motors (CIAM) (Russia)
- PANKOV Sergey – Central Institute of Aviation Motors (CIAM) (Russia)

A Numerical Analysis on the Aerodynamic Noise of Cross-FLow Fan by Using a 2-D Urans Simulation and Acoustic Analogy

- LEE Myungsung – Sogang University (South Korea)
- KANG Seongwon – Sogang University (South Korea)
- HUR Nahmkeon – Sogang University (South Korea)
- PARK Jeongtaek – LG Electronics (South Korea)

Thursday 19 April 2012

D1 Computational Aeroacoustics

Influence of Blade Number on Aerodynamic Noise of Propeller Fans for Outdoor Unit of Air-Conditioner

- IWASE Taku – Hitachi, Ltd. Hitachi Research Laboratory (Japan)
- KISHITANI Tetsushi – Hitachi Appliances, Inc. (Japan)
- FURUKAWA Masato – Department of Mechanical Engineering, Kyushu University (Japan)

Optimization of Axial Fans with Highly Swept Blades with Respect to Losses and Noise Reduction

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

Stationary and Moving Surface Ffowcs Williams and Hawkings Computations of an Isolated Radial Impeller

- HEINEMANN Till – University Erlangen-Nuremberg (Germany)
- SCHEIT Christoph – University Erlangen-Nuremberg (Germany)
- SPRINGER Matthias – University Erlangen-Nuremberg (Germany)
- MÜLLER Stefan – University Erlangen-Nuremberg (Germany)
- BECKER Stefan – Institute of Process Machinery and Systems Engineering, University Erlangen-Nuremberg (Germany)

Numerical Method for 3D Computation of Turbomachinery Tone Noise

- ROSSIKHIN Anton – Central Institute of Aviation Motors (CIAM) (Russia)
- BRAILKO Igor – Central Institute of Aviation Motors (CIAM) (Russia)
- PANKOV Sergey – Central Institute of Aviation Motors (CIAM) (Russia)
- MILESHIN Victor – Central Institute of Aviation Motors (CIAM) (Russia)

D2 Aerodynamic Design

Evaluation of Squirrel-Cage Fans for HVAC Applications in Public Transport: Key Parameters and Design Guidelines

- VELARDE-SUAREZ Sandra – Universidad de Oviedo (Spain)
- GUERRAS-COLON F. Israel – Universidad de Oviedo (Spain)
- BALLESTEROS-TAJADURA Rafael – Universidad del Oviedo (Spain)
- GONZALEZ Jose – Universidad de Oviedo (Spain)
- ARGÜELLES DIAZ Katia M. – Universidad de Oviedo (Spain)
- FERNANDEZ ORO Jesus M. – Universidad de Oviedo (Spain)
- SANTOLARIA MORROS Carlos – Universidad de Oviedo (Spain)

A Design of Experiment for Evaluating Installation Effects and the Influence of Blade Loading on the Aeroacoustics of an Automotive Engine Cooling Fan

- TANNOURY Elias – Valeo Thermal Systems (France)
- DEMORY Bruno – Valeo Thermal Systems (France)
- HENNER Manuel – Valeo Thermal Systems (France)
- BONNET Pierre-Alain – Valeo Thermal Systems (France)
- CAULE Patrice – Technofan (France)
- CRÉTEUR Yoann – Université de Poitiers (France)

Design and Aerodynamic Performance of High Pressure Axial Flow Fan

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

Experimental Investigation of Ducted Counter-Rotating Axial Flow Fans

- NOURI Hussain – Arts et Métiers ParisTech - Laboratoire DynFluid (France)
- RAVELET Florent – Arts et Métiers ParisTech - Laboratoire DynFluid (France)
- BAKIR Farid – Arts et Métiers ParisTech - Laboratoire DynFluid (France)
- SARRAF Christophe – Arts et Métiers ParisTech - Laboratoire DynFluid (France)

D3 Design and Optimisation

CFD Simulations to Predict the Energy Efficiency of an Axial Fan for Various Casing Configurations

- GUEDEL Alain – CETIAT (France)
- ROBITU Mirela – CETIAT (France)
- CHAULET Vivian – Ziehl-Abegg FMV (France)

CFD Models for the Analysis of Rotor-Only Industrial Axial-Flow Fans

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

Friday 20 April 2012

E1 Noise of Cooling Fans in Automotive Applications

Acoustic Characterization of a Centrifugal Fan in an Automotive Ventilation Unit Sound Predicting Model

- AL MEZZAWI Ali – CETIM & UTC (France)
- GOTH Yvon – CETIM (France)
- CARNIEL Xavier – CETIM (France)
- CHERIAUX Olivier – Valeo (France)

Cetim_Ventil: a Software to Predict the Noise of a Cooling Fan under a Hood

- GOTH Yvon – CETIM (France)

Aeroacoustical Investigations on Axial Fans for Automotive Cooling Systems

- ZAYANI Mohamed – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
- ÇAĞLAR Şaban – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
- GABI Martin – Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)

Noise Control for Cooling Fans on Heavy Vehicles

- ALLAM Sabry – KTH - Competence Centre for Gas Exchange (CCGEx) (Sweden)
- ÅBOM Mats – KTH - The Marcus Wallenberg Laboratory (MWL) (Sweden)
- WESTER Lars – Sontech AB (Sweden)

E2 Codes, Standards and Legislation

An Initial Assessment of the Changes that will Appear in a Forthcoming (2012) Revision of EN 12101-3

- SHEARD Geoff – Fläkt Woods (United Kingdom)
- JONES Neil – NMJ Consultancy (United Kingdom)

Investigations on the Swirl Flow Caused by Axial Fans - A Contribution to the Revision of ISO 5801

- MATTERN Philipp – Karlsruhe Institute of Technology (Germany)
- SIEBER Sten – Karlsruhe Institute of Technology (Germany)
- ČANTRAK Đorđe – University of Belgrade - Faculty of Mechanical Engineering (Serbia)
- FRÖHLIG Friedrich – Karlsruhe Institute of Technology (Germany)
- ÇAĞLAR Şaban – Karlsruhe Institute of Technology (Germany)
- GABI Martin – Karlsruhe Institute of Technology (Germany)

E3 Flow Characteristic of Centrifugal Fans

Flow Characteristics of Backward Curved Centrifugal Fan with Rectangular Casing

- HAYASHI Hidechito – Nagasaki University (Japan)
- NAKAMURA Kyohei – Nagasaki University (Japan)
- SASAKI Souichi – Nagasaki University (Japan)
- SHIRAHAMA Seiji – Panasonic Ecology Systems Co. Ltd (Japan)
- NAGATA Atsushi – Panasonic Ecology Systems Co.Ltd (Japan)

Toward the CFD Simulation of Sirocco Fans: from Selecting a Turburlence Model to the Role of Celle Shapes

- DARVISH Manoochehr – University of Applied Sciences - HTW Berlin (Germany)
- FRANK Stefan – University of Applied Sciences - HTW Berlin (Germany)

Effects of Housing Geometry on the Performance and Noise of a Two-Outlet Centrifugal Fan - A Numerical Study

- WONG Ian Y. W. – The Hong Kong Polytechnic University (China)
- LEUNG Randolph C. K. – The Hong Kong Polytechnic University (China)
- LAW Anthony K. Y. – Raymond Industrial Limited (China)

F1 Noise Prediction and Investigation for Axial Fans

Free and Scattered Acoustic Field Predictions of the Broadband Noise Generated by a Low-Speed Axial Fan

- KUCUKCOSKUN Korcan – Ecole Centrale de Lyon (France)
- CHRISTOPHE Julien – von Karman Institute for Fluid Dynamics (Belgium)
- SCHRAM Christophe – von Karman Institute for Fluid Dynamics (Belgium)
- TOURNOUR Michel – LMS International (Belgium)

Broadband Noise Modelling and Prediction for Axial Fans

- DE GENNARO Michele – AIT Austrian Institute of Technology G.m.b.H. (Austria)
- KUEHNELT Helmut – AIT Austrian Institute of Technology G.m.b.H. (Austria)

Vortex Generation Behind the Cylinder Cascade of Fan Grill

- MILAVEC Matej – Hidria Institut Klima d.o.o. (Slovenia)
- SCHWEIGER Vlado – Hidria Institut Klima d.o.o. (Slovenia)
- HOČEVAR Marko – University of Ljubljana, Faculty for Mechanical Engineering (Slovenia)
- EBERLINC Matjaž – University of Ljubljana, Faculty for Mechanical Engineering (Slovenia)
- ŠIROK Branko – niversity of Ljubljana, Faculty for Mechanical Engineering (Slovenia)

F2 Vibrations and Failure Analysis

Fin Fan Vibration Remediation

- BALDWIN Richard M. – Southwest Research Institute (United States)
- PANTERMUEHL P. Joe – Southwest Research Institute, retired (United States)
- PILLAI Venkat K. – Kuwait National Petroleum Company (Kuwait)

Failure Analysis of Impeller Blade Cracking in a Mechanical Vapour Recompressor (MVR) Fan

- DONOHUE Brian – University of Canterbury (New Zealand)
- KRAL Milo – University of Canterbury (New Zealand)

F3 Active Noise Control & Noise Analysis

25 Years Ago, Active Control First Appeared as a Great Solution for Fan Noise - What has Happened?

- WISE Steve – Wise Associates (United States)

Implementation of Active Noise Control into Different Fan Applications

- SCHNEIDER Marc – ebm-papst Mulfingen GmbH & Co. KG (Germany)
- BALBACH Ralf – ebm-papst Mulfingen GmbH & Co. KG (Germany)

Fan Noise Analysis Using Microphone Array

- MINCK Olivier – MicrodB (France)
- BINDER Nicolas – ISAE (France)
- CHERRIER Olivier – ISAE (France)
- LAMOTTE Lucille – MicrodB (France)
- BUDINGER Valérie – ISAE (France)

ABSTRACTS

NOISE PROVES NOTHING - SOURCES OF FAN NOISE AND THEIR PREDICTION

Thomas CAROLUS / University of Siegen (Germany)

Thomas CAROLUS, Professor of Applied Fluid Mechanics & Turbomachinery, University of Siegen, Germany

Biography: Thomas is well known for his experience with fan and blower aerodynamics and aeroacoustics and turbomachinery systems. His book on these topics is widely used in academia and industry. He is a Visiting Professor in England, USA, France, Laos and Albania and serves as a reviewer for numerous journals and conferences.

Prior to his position at the University, Thomas was Senior Engineer at Bosch GmbH in charge of heating, ventilation and air conditioning components for automotive application. He graduated from Karlsruhe Institute of Technology and the Georgia Institute of Technology, receiving his PhD from the former.

thomas.carolus@uni-siegen.de

INLET FLOW DISTORTION AND LIP SEPARATION IN DUCTED FANS USED IN VTOL UAV SYSTEMS

Cengiz CAMCI / Penn State University (United States of America)
Ali AKTURK / Siemens Energy (United States of America)

This paper describes a novel ducted fan inlet flow conditioning concept that will significantly improve the performance and controllability of ducted fan systems operating at high angle of attack. High angle of attack operation of ducted fans is very common in VTOL (vertical take off and landing) UAV systems. The new concept that will significantly reduce the inlet lip separation related performance penalties in the edgewise/ forward flight zone is named “DOUBLE DUCTED FAN” (DDF). The current concept uses a secondary stationary duct system to control “inlet lip separation” related momentum deficit at the inlet of the fan rotor occurring at elevated edgewise flight velocities. The DDF is self-adjusting in a wide edgewise flight velocity range and its corrective aerodynamic effect becomes mordde pronounced with increasing flight velocity due to its inherent design properties. Most axial flow fans are designed for an axial inlet flow with zero or minimal inlet flow distortion. The DDF concept is proven to be an effective way of dealing with inlet flow distortions occurring near the lip section of any axial flow fan system, especially at high angle of attack. In this present paper, a conventional baseline duct without any lip separation control feature is compared to two different double ducted fans named DDF CASE-A and DDF CASE-B via 3D, viscous and turbulent flow computational analysis. Both hover and edgewise flight conditions are considered. Significant relative improvements from DDF CASE-A and DDF CASE-B are in the areas of vertical force (thrust) enhancement, nose-up pitching moment control and recovery of fan through-flow mass flow rate in a wide horizontal flight range.

cxc11@psu.edu

Cengiz Camci, professor of aerospace engineering, Pennsylvania State University.

Cengiz has worked at Pennsylvania State University since 1986 and currently directs his research activities in his "Turbomachinery Aero-heat Laboratory". Following his doctoral degree (awarded by the Von Karman Institute for Fluid Dynamics) in 1985, Cengiz has been an invited lecturer at the VKI Lecture Series and a mission consultant at the ATO/AGARD RTO Research Technology Organisation. His research interests include aerothermodynamics of turbomachinery, analytical and experimental fluid dynamics, heat transfer in air breathing propulsion systems, turbine cooling, ducted fan aerodynamics for VTOL UAV systems and san erosion of helicopter rotor blade tips.
Cengiz has received three best paper awards from the ASME Heat Transfer Division, Fluids Engineering Division and IGTI Education Committee throughout his career and became a fellow of ASME in 2007. He completed many research programmes sponsored by NSF, NASA, DOE, AGTSR, General Electric (GE), Pratt&Whitney (P&W), Solar Turbines, Siemens Energy, Sikorsky Aircraft Company (SAC) and United Technologies Research Center (UTRC). He has also published more that 140 papers in his research areas.

EU POLICIES ON ENERGY EFFICIENCY AND THE IMPACT ON FAN MANUFACTURERS

Guido DE WILT / Policy Officer - Energy Efficiency - European Commission
(Directorate - General for Energy) (Belgium)

Guido de Wilt , Policy Officer – Energy Efficiency, European Commission, Belgium

Biography : Guido is a policy officer in the team responsible for the efficiency of energy-related products. He is working on legislation for ecodesign minimum requirements and energy labelling for industrial products such as fans and pumps, as well as for heating products. He is also responsible for the work on standardisation of the unit.

He joined the European Commission in 2003 to work in the field of energy efficiency, starting as a policy officer responsible for developing and implementing legislation and state aid guidelines for high-efficiency cogeneration. He has also worked on existing energy efficiency legislation on lighting and boilers and contributed to new legislation on energy services and energy efficiency in buildings. Later his responsibilities shifted towards energy efficiency in international relations and standardisation.

Guido graduated from Eindhoven University of Technology and went on to work as a lecturer there in the field of ceramics and thermodynamics. He also lectured in physics and environmental science at Brabant Polytechnic.

Wilhelmus.DE-WILT@ec.europa.eu

On the role of CFD in the aerodynamic development of industrial fans or from equations to colors (and back again)

Alessandro CORSINI / Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)

The current industrial practice entails a number of codes for fluid dynamics simulations, and this circumstance let the aerodynamic designers choose among different methodologies to transform governing equations into useful flow field description.

The selection of the proper numerical approach can be quite a conundrum for engineers, as the number of available options is so high that ending up with just stunning coloured pictures is more than a possibility.

In the following some of the possible choices for the numerical investigation of fans and other turbomachinery components are discussed, spanning from RANS to LES computations and dealing with different aspects such as virtual prototyping, performance prediction, noise suppression and three- dimensional design.

corsini@dma.ing.uniroma1.it

Alessandro CORSINI, Associate Professor of Fluid Machinery, Sapienza University of Rome

Alessandro is the leader of the team for industrial fans and blowers CFD group at the Department of Mechanical and Aerospace Engineering, Sapienza University, Rome. Following his PhD in 1996 on numerical methods for fluid-dynamics, he has worked in this field by developing original stabilized numerical formulations for the computation of turbulent flows in turbomachinery. He was invited ERASMUS- SOCRATES lecturer at Budapest University of Technology and Economics and, since 2004, he is visiting scientist of the Team for Advanced Flow Simulation and Modeling TAFSM, Rice University in Houston, TX USA. In the last decade, Alessandro's research interests focused more on the development of fan technologies and include passive noise control devices, stall aerodynamics and early-warning diagnostics, and modeling of the erosion in induced draft fans. Alessandro is the author of more than 100 technical papers in his research areas and he is also the author of 3 international patents in turbomachinery applications.

FAN TONAL NOISE REDUCTION USING CALIBRATED OBSTRUCTIONS IN THE FLOW: AN EXPERIMENTAL APPROACH

Yvon GOTH / CETIM (France)
Michel BESOMBES / CETIM (France)
Christian CHASSAIGNON / DYVA (France)
Anthony GERARD / Université de Sherbrooke (Canada)

We present the experimental results of a technology aiming at the cancellation of the tonal noise of fans, using calibrated obstructions placed upstream or downstream in the flow close to the fan. When placed at the optimal distance and angle, the obstruction generates pressure pulsations that interact with the fan pressure fluctuations in a destructive way, allowing to cancel the noise at the blade passing frequency, and, in some cases, at its first harmonic.

A parametric study has been conducted on two types of fans, axial and centrifugal. Parameters that have been studied are the obstructions shape and position and the fan speed. More than 50 different configurations have been investigated. A robot has been used to position the obstacles, allowing to get sound maps in distance and angle coordinates for each configuration.

yvon.goth@cetim.fr

NUMERICAL OPTIMIZATION OF THE TONAL NOISE OF A BACKWARD CENTRIFUGAL FAN USING A FLOW OBSTRUCTION

Franck PEROT / Exa Corporation (United States of America)
Min-Suk KIM / Exa Corporation (United States of America)
Vincent LE GOFF / EUROXA (France)
Xavier CARNIEL / CETIM (France)
Yvon GOTH / CETIM (France)
Christian CHASSAIGNON / DYVA (France)

Broadband and particularly tonal content of the noise radiated by fans can be perceived as a strong annoyance for people staying in their vicinity and can affect working conditions of operators. As a consequence, fan noise reduction represents a real challenge and an important business criterion for fan and integrated system suppliers. The origin of tonal noise is mainly related to the uniformity and unsteadiness of the inlet flow and to rotor-casing interactions and occurs at the Blade Passing Frequency (BPF) f0 and harmonics 2.f0, 4.f0,… which frequency values depend on the rotation frequency and the number of blades of the fan.

In this study, our interest is focused on a passive noise control device so-called flow obstruction significantly reducing the fan BPF noise. This device is calibrated meaning that its shape, position and orientation with respect to the fan are optimized. The obstruction is located upstream of a centrifugal fan with a heat exchanger placed in between is order to load the fan as in real hood conditions. Some measurements performed at various operating conditions on the CETIM fan aeroacoustics bench test show that for an optimal position and orientation of the obstruction, the BPF noise is reduced by 10-13 dB. These experiments will be presented and detailed in the final version of the paper.

The goal of this paper is to use a CFD/CAA approach for modeling this problem and numerically determine the optimized position of the obstruction. Simulations results are also used to provide a better understanding of the physical mechanisms involved in the noise reduction. The CFD/CAA code PowerFLOW 4.3 based on the Lattice Boltzmann Method (LBM) is used. This flow and acoustics solver is transient, explicit and compressible and the full fan and bench geometries are included in the simulation domain. The fan is truly rotating during the simulation using a Local Reference Frame (LRF) approach. The flow-induced noise contribution radiated from the turbulent flow is simulated during the transient flow simulation and no coupling to another acoustics propagator solver is required. Simulations are carried out at two operating conditions with and without the obstructions and the noise reduction from the obstruction recovered. Additional post-processing of the transient flow and acoustics fields show the effect of the obstruction on the radiated noise and will be presented in the final paper.

perot@exa.com

TONAL NOISE CONTROL FROM CENTRIFUGAL FANS USING FLOW CONTROL OBSTRUCTIONS

Anthony GERARD / Université de Sherbrooke (Canada)
Michel BESOMBES / CETIM (France)
Alain BERRY / Université de Sherbrooke (Canada)
Patrice MASSON / Université de Sherbrooke (Canada)
Stephane MOREAU / Université de Sherbrooke (Canada)

Tonal noise originates from non-uniform flow that causes circumferentially varying blade forces and gives rise to a considerably larger radiated dipolar sound at the blade passage frequency (BPF) and its harmonics. The approach presented in this paper adapts a method previously developed for axial fans to control tonal noise using obstructions in the flow to destructively interfere with the primary non-uniform flow arising from stator/rotor interaction.

To target the circumferential modes of the unsteady forces to be controlled, the analytical model of tonal noise radiation from centrifugal fans in free field developed by Khelladi et al. is first analysed. Three major circumferential modes of the rotor unsteady forces radiate tonal noise at BPF: mainly the circumferential B-1 and B+1 orders for radial and tangential forces and the Bth order for the axial forces, where B is the number of blades. Controlling one of these modes can lead to significant BPF tone attenuation. The flow control obstruction is located such that the secondary radiated noise is of equal magnitude but opposite in phase compared to the primary noise. The magnitude of the secondary noise is controlled by the axial distance between the rotor and the obstruction or the size of the lobes of the obstruction. The phase of the secondary noise is controlled by the angular position of the control obstruction.

Experiments were carried out for a centrifugal fan test bench to validate the method for controlling BPF tonal noise by carefully positioning obstructions in a duct in the upstream flow.

anthony.gerard@usherbrooke.ca

Innovative Fan Drives for Hybrid Trucks

Thomas JUSTIN / Volvo 3P (France)
Tessa MORGAN / Volvo 3P (France)
Julien BRALY / Volvo 3P (France)

With ever-increasing fuel prices and forthcoming stringent regulations on the reduction of CO2 emissions, the desire for better fuel economy in the truck sector continues to drive towards an optimisation of energy use. This leads to a need to reduce parasitic losses on the vehicle, including those from the engine accessory drive. Accessory drive losses account for a significant proportion of the engine power and have a direct impact on vehicle fuel consumption.

An additional driver for an evolution in accessory drives is the development of Hybrid Electric Vehicles, where there is a need for accessories to function even when the internal combustion engine is stopped.

This paper addresses, in particular, the need for an evolution of the main engine mounted cooling fan drive, for trucks. Typically, in hybrid vehicles the cooling of the electrical components of the powertrain, such as the energy storage systems, power converters and electric machines, is assured by an auxiliary cooling system. Auxiliary electrically driven fans are used to meet cooling needs in hybrid or electric drive modes, adding cost, weight and additional packaging constraints to the vehicle.

This paper explores the possibilities for using the main engine cooling fan to provide cooling air flow even when the internal combustion engine is stopped. The required functionalities, including fan speed modulation, fan acceleration, noise levels and efficiency in several operating conditions, and fail safe modes, are established and prioritised. Different solutions for the engine cooling fan drive are discussed. The development of an innovative electro-mechanical solution is presented: The working principle, modeling, concept choices and mechanical and electrical dimensioning to reach the required functionalities are detailed.

Finally, the conclusions summarise the main challenges in developing such an electro-mechanical fan drive, as well as the advantages and disadvantages compared to traditional fan drives for both hybrid and non-hybrid applications.

thomas.justin@volvo.com

DESIGNING A CENTRIFUGAL FAN OF CARBON FIBER LAMINATE FOR HIGH CIRCUMFERENTIAL SPEED

Heiko RATTER / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
Herrmann MAUCH / Hürner-Funken GmbH (Germany)
Benjamin HANGS / Fraunhofer-Institude for Chemical Technology (Germany)
Şaban ÇAĞLAR / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
Martin GABI / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)

Centrifugal fans made of plastic are commonly used in systems where the transported fluids containing acid materials. Nonmetallic materials have, additionally to their persistence against corrosion, the advantage of a low noise level. Conversely the low strength of plastic limits the load to the structure and hence the maximum circumferential velocity.

Within the scope of this project a given impeller design of plastic is optimized with the goal of higher efficiency and the ability of higher circumferential speed.

As material for the new fan carbon fiber laminate is chosen, due to its resistance against high stresses and persistence to acid materials. Because carbon fibers must be positioned without any sharp bends to avoid tension cracks, the use of such fiber materials restricts the design options of the fan. These requirements lead to a non-standard geometry especially at the intersections blade to shroud and blade to hub.

This paper presents the approach of designing an impeller geometry which fulfills all required restrictions. With the goal to construct a high efficient fan, 3D steady and unsteady CFD calculations of the impeller and the complete fan are carried out. They are based on a block structured mesh of 1.3 million cells for the impeller and inlet nozzle resp. 2.8 million cells for the complete fan.

21 different fan geometries were calculated, successively optimizing the impeller geometry. The calculations of the final impeller show higher efficiency rates at all given operation points, additionally the geometry fulfills all restrictions given by the material.

ratter@kit.edu

USE OF GRADE 80 STEEL IN FABRICATED CENTRIFUGAL IMPELLER

Nicholas BRUNCK / American Fan Company (United States of America)
Paul OKELEY / Fläkt Fan Group (United States of America)
Geoff SHEARD / Fläktwood Ltd (United Kingdom)

The use of Grade 80 Steel is evaluated for application in the construction of backward curved centrifugal impellers. Typical construction of fabricated centrifugal impellers incorporates an intermediate shroud in order to attain higher tip speeds, however, the authors examined the use of other higher strength materials to attain the same tip speed without the need for an intermediate shroud. The aerodynamic advantage of removing the ring was known from historical test data and was confirmed by direct testing. In the case examined by the authors, a duty point of 4.6 m3/sec at 3363 Pa was required at a total efficiency of 72.7%. The space envelope allowed for a 508 mm diameter impeller. An existing pair of 381 mm diameter impellers was available for testing the intermediate shroud's effects at lower speeds.

The test results, scaled via fan laws, showed that without the intermediate shroud, the 508 mm diameter impeller could reach the desired flow and pressure duty point as well as efficiency, while the impeller with the intermediate shroud could meet the efficiency requirement at the flow and pressure duty point. Finite element analysis confirmed the Grade 80 material was appropriate for the speed required. Additionally, finite element analysis confirmed that the natural frequencies of the new impeller compared favorably with those of the original impeller. Despite a higher cost of material, the elimination of welding the intermediate shroud resulted in a cost savings in excess of 40%. In summary, the use of Grade 80 Steel allows for lower costs in some high speed applications while improving fan efficiency. Grade 80 Steel could be considered as an alternative to Grade 50 Steel and other lower strength materials in demanding applications where high impeller speed and high efficiency are primary requirements.

nick.brunck@flaktwoods.com

MAXIMUM ACHIEVABLE EFFICIENCY OF CENTRIFUGAL FANS WITHOUT HOUSING

Yingan XIA / Punker GmbH (Germany)
Şaban ÇAĞLAR / Karlsruhe Institut für Technologie (Germany)
Heiko RATTER / Karlsruhe Institut für Technologie (Germany)
Martin GABI / Karlsruhe Intitute für Technologie (Germany)

The present paper is focused on centrifugal fan wheels with backward curved blades and a rotating diffuser in free-blowing operating (without housing). Moreover, the consideration is confined to the flow area within impeller.

Such centrifugal fans are very popular for the use in air handling units because of advantages such as compact design and direct drive etc.. The fan efficiency ηr is, however, a little lower than the fan with housing, which converts the dynamic flow energy of the air directly into usable static flow energy. There is no doubt a further potential to improve the efficiency through a further aerodynamic optimization of the wheel geometry. It is, however, exciting to know on which parameters the efficiency of these fans ηr primarily depends and what is the theoretic limit of the efficiency maximum achievable ηr,max.

The basis for the theoretic consideration is the Euler's main equation, which describes the energy conversion in the fan wheel. For an idealized flow within the fan wheel the relation between the fan efficiency ηr and the hydraulic efficiency of the blading (incl. inflow and outflow from the wheel) ηhydr is derived in function of the other technical operating and geometric parameters.

A theoretic estimation of the expected efficiency ηr of an aerodynamically "lossfree" fan can be carried out by ηhydr = 1. Inversely from the really measured/achieved efficiency ηr of a fan, the theoretic, hydraulic efficiency ηhydr can then be estimated as to the introduced formula aiming at evaluating the potential for a realistic improvement.

Three wheel examples with different blade geometries show how much the efficiency of a fan without housing depends on the working point (air volume and pressure) apart from the blade geometry. The limit of the maximum efficiency achievable of a centrifugal fan should in fact be higher than the real value of 73% presently achieved at the working point with a dimensionless volume number: 0.26 and a dimensionless pressure number: 0.57. The margin for a further improvement through an aerodynamic optimization of the flow in the wheel should, however, be highly limited for this wheel due to the high value already achieved.

In order to increase the total efficiency of the fan, the efficiency of other components like motor, control, etc. has to be improved as well.

xia@punker.de

DESIGN IMPROVEMENTS OF SIROCCO TYPE FANS BY MEANS OF COMPUTATIONAL FLUID DYNAMICS AND STEREO PARTICLE IMAGE VELOCIMETRY

Stefan FRANK / University of Applied Sciences HTW Berlin (Germany)
Manoochehr DARVISH / University of Applied Sciences HTW Berlin (Germany)
Bastian TIETJEN / University of Applied Sciences HTW Berlin (Germany)

The performance of Sirocco type fans can be tested on conventional chamber test rigs with low uncertainties concerning flow-rate, pressure rise and efficiency. However, the conventional optimization of these systems require a lot of experimental effort and is very time consuming. Hence, optimization by numerical means such as Computational Fluid Dynamics would be favourable. In order to do so a reliable data base for the assessment of numerical results has to be provided. 2D Particle Image Velocimetry measurements can be performed with relatively low experimental effort. Comparison between velocity fields from experiment and simulation are already quite consistent. However, the third, out-of-plane velocity component remains undetected in principle. Hence, Stereoscopic Particle Image Velocimetry is applied that provides instantaneous 3D velocity fields in the measurement plane. Thus, besides the main flow the secondary vortex flow can be detected and the overall quality of measurement increases significantly. However, this is at the cost of a higher experimental effort. Furthermore, for this kind of internal flow not the whole velocity field of interest can be evaluated due to the shadowing of the scroll, the tongue and the rotor. Nevertheless, data is obtained that could not be achieved otherwise.

Numerical calculations are performed using steady RANS equations with SST turbulence model and highly resolved grids. Both structured and unstructured grids with hexahedral and polyhedral cells respectively can be applied. For a cell based solver and a comparable number of elements, both mesh types will yield equally good results at a similar numerical effort. However, mesh generation is much more flexible and significantly faster with unstructured grids. Comparison between numerical and experimental data shows deviations of less than five per cent for pressure rise and efficiency for a wide operating range. Direct and point wise comparison of the velocity fields show a high degree of consistency between the two methods. Once the numerical method has been established and proved to be accurate it can be used for the virtual optimization of e.g. Sirocco fans before building prototypes. Starting from a previous design an optimized design variant could be developed, that has a considerably smaller size and yields a much higher static efficiency at the best efficiency point. Therefore, Computational Fluid Dynamics in conjunction with Particle Image Velocimetry is a useful way to reliably improve Sirocco type fans.

stefan.frank@htw-berlin.de

FLUID DYNAMIC DESIGN AND OPTIMIZATION OF A DOUBLE ENTRY FAN DRIVEN BY TRACTOR POWER TAKE OFF FOR MIST SPRAYER APPLICATION

Michele PINELLI / University of Ferrara (Italy)
Mirko MORINI / University of Ferrara (Italy)
Mauro ROSSINI / Ideal Srl (Italy)
Cristian FERRARI / University of Ferrara (Italy)

Mist sprayers are commonly used in conventional agriculture for the distribution of herbicides, pesticides and fertilizers. For this purpose in particular, truck-mounted mist sprayers are used to spray insecticides to trees, shrubs, and tall grass in rural areas and to road borders and courtyards in urban areas.

The coupling of a fan, used to supply air to a mist sprayer, and a IC engine is problematic since, in this case, for the correct design of the fan it is not possible to define a priori the operating point. In fact, the rotational speed is not fixed as in the case of an electric motor driven fan, but is determined as an equilibrium of the power supplied by the engine and the power absorbed by the fan to recover the pressure drops of the mist sprayer system.

In this paper, the fluid dynamic design of a double entry fan driven by tractor power take off for mist sprayer applications is presented. The design is carried out by means of an integrated 1D/3D numerical procedure based on the use of CFD simulations.

Moreover, an experimental campaign performed to characterize the existent fans is presented. The collected data are elaborated in order to fix a design starting point and to validate the CFD procedure.

The CFD simulations are then used either at the preliminary design stage to choose among competitive one- or two-dimensional geometries and then to test the generated three-dimensional geometries. The results show how the different design choices could impact on the performance parameters and, finally, how the analysis of the various alternatives allows the determination of the overall geometry of a complete and performing centrifugal fan.

Finally, the double entry fan has been prototyped and tested. The test confirmed that the designed fan met the manufacture requests in terms of efficiency, maximum flow rate, head and noise level.

michele.pinelli@unife.it

COMPUTATIONAL AEROACOUSTIC ANALYSIS OF AN INDUSTRIAL FAN APPLICATION USING A HYBRID APPROACH

Marie CABROL / Free Field Technologies (Belgium)
Romain LENEVEU / Free Field Technologies (Belgium)
Diego D'UDEKEM / Free Field Technologies (Belgium)
Julien MANERA / Free Field Technologies (Belgium)
Yves DETANDT / Free Field Technologies (Belgium)
Fred MENDONCA / CD-Adapco (United Kingdom)

Fans are used in many domains: ventilation, air conditioning, cooling systems for electric and thermal motors, electrical domestic, power generation in industrial compressors and turbines…With the growth of the acoustic comfort demand coupled to a more restrictive legislation, the new fan design are constrained to challenging noise control and reduction. Flow noise generated by a fan is often one of the major sources of noise.

Among the different techniques currently available to address this type of problem, a hybrid approach is recommended for industrial applications. In this approach, the unsteady flow field is first determined by a CFD solver like STAR-CCM+, and in a second stage the noise sources are propagated by a finite element acoustic software like ACTRAN. This allows clear identification of noise sources including those highly amplified during the acoustic propagation process. For many reasons, fan noise prediction using such approach is challenging and requires some advanced investigations. On the CFD side, acoustical non-reflective treatments and the manipulation of rotating-static domains to predict the unsteady compressible flow is complex and requires the use of advanced interfaces between static and rotating parts. On the acoustic side, the computation of moving CAA sources into the frequency domain requires the use of specific non-reflecting boundary conditions and a correct handling of the noise source and the acoustic propagation in a non-uniform medium.

In the first part of the present paper, the CFD model is described, highlighting the special features required for an efficient and accurate flow modeling inside a fan configuration. In a second step, the acoustic modeling technique is presented. This technique post-process the unsteady CFD results and computes the different excitations provided to the acoustic model. The acoustic model is described, focusing on the boundary conditions, propagation model in a non-uniform medium. In particular, the effect of a non-uniform flow field in the diffuser and in the outlet duct will be addressed by comparing the results of Lighthill and Möhring analogy. In both parts, best practices will be formulated and recommended. Some extensions involving aero-vibro acoustic concepts (aero acoustic sources exciting the fan casing and inducing noise in the external domain are presented.

The paper illustrates the application of a hybrid approach for fan noise modeling. The computational cost of the simulation is compared to the theoretical cost of a full direct simulation involving flow and acoustic simulation.

mca@temp.fft.be

Experiments and Numerical Predictions of Flow Rates and Aero-Acoustics from Small Radial Notebook Blowers

Jessica GULLBRAND / Intel Corporation (United States of America)
Willem BELTMAN / Intel Corporation (United States of America)

The performance of a notebook system depends upon the thermal cooling capacity of the blower/heat exchanger combination, which in turn is dictated by the effective airflow rate produced by the blower. The cooling capacity increases with increased airflow, but so does the acoustic noise. However, ergonomic limits of the acoustic noise from notebooks need to be adhered to, which causes most notebook blowers to operate below maximum rotational speeds. This leads to a scenario in which the performance of many notebook systems is acoustically limited. It is therefore of great importance to optimize notebook blower designs for maximum flow rate and low acoustic noise. This optimization has historically been performed by building expensive prototypes and performing experimental investigations. It is beneficial to numerically optimize the blower designs to reduce design time and prototype cost, and this requires accurate numerical techniques. It is only recently that numerical characterization of airflow and acoustic performance has received increased attention due to the availability of advanced simulation techniques in commercial software. The accuracy of these techniques to predict noise generated by small radial blowers need to be determined.

In this paper, numerical simulations of a typical radial notebook blower are performed to first determine accuracy of the airflow and pressure performance of the blower when compared to experimental data. The flow field simulations are performed using unsteady Reynolds-Averaged Navier-Stokes (URANS) equations. Thereafter, aero-acoustic simulations are undertaken. Two different blower setups are investigated: one setup where the blower is in free field and the other one where the blower is mounted in a bracket. The bracket is generally used in experimental evaluation of notebook blowers. The Ffowcs-Williams and Hawkings (FWH) analogy is used in the aero-acoustic predictions. Good agreement is demonstrated for airflow and pressure performance between measurements and numerical predictions. The acoustic experiments show good repeatability for the blade-pass-frequency (BPF), while large variability is observed for the second harmonic. The noise level at the BPF is the highest peak value and the predictions show promise, but accurate aero-acoustic predictions are still challenging.

jessica.gullbrand@intel.com

Optimization of Fan Noise by Coupling 3D Inverse Design and Automatic Optimizer

Mehrdad ZANGENEH / University College London (United Kingdom)
Matthieu DE MAILLARD / Advanced Design Technology Ltd (United Kingdom)

A methodology for designing fans to meet multi-objective design criteria is presented. The method combines a 3D inviscid inverse design method with a multi-objective genetic algorithm to design fans which meet various aerodynamic aero-acoustics and geometrical requirements. The parameterization of the blade shape through the blade loading enables 3D optimization with very few design parameters. A generic fan stage is used to demonstrate the proposed methodology. The main design objectives are improving profile losses to improve design point efficiency and reduce diffusion for better stall performance. The optimization is performed subject to certain constraints on Euler head, throat area, thickness and meridional shape so that the resulting fan stage can meet both design and off-design conditions. A Pareto Front is generated for the two objective functions and 3 different configurations on the Pareto front are selected for detailed study by 3D RANS code. The CFD results confirm the main outcomes of the optimization process.

m.zangeneh@ucl.ac.uk

FAN-SYSTEM INTERACTION AND BLOCKAGE EFFECTS FOR HVAC MULTI-FAN UNITS

Sandra VELARDE-SUAREZ / Universidad de Oviedo (Spain)
F. Israel GUERRAS-COLON / Universidad de Oviedo (Spain)
Jose GONZALEZ / Universidad de Oviedo (Spain)
Katia M. ARGÜELLES DIAZ / Universidad de Oviedo (Spain)
Jesus M. FERNANDEZ ORO / Universidad de Oviedo (Spain)
Carlos SANTOLARIA MORROS / Universidad de Oviedo (Spain)

Small fans, both axial and centrifugal, are often installed in the air conditioning systems for public transportation vehicles. These fans are driven at relatively high speeds (around 4000 rpm might be reached), with extremely variable working conditions, probably far from the nominal ones. All these facts, together with the need to reduce fabrication costs, give rise to frequent instability problems, low performance, low efficiency and high levels of vibrations and noise.

Besides, when using multi-fan units, there are important geometrical restrictions and the interactions with the system play an increasingly important role. In the evaporator units of any of the before mentioned vehicles it is common the use of parallel configurations of multi-fan units. In such units, the modern designs restrict the volume of air available around the fan and closer location of the units produce important interaction problems.

The main goal of this work is the optimization of a multi-fan unit, finding the minimum distance between fans to reduce within a reasonable limit the interaction effects, with other fans and with the ventilation system. For such objective, an experimental set-up is built and several experiments have been carried out, particularly measurements to find the fan-wall interaction, the wall-fan-wall interaction and the interaction among three fans have been performed. Rotational speed, power consumption and noise measurements were performed for two different flow rates, by regulating the opening valve in the conduct.

From the experiments, the conclusion is that the minimum distance between any fan and the wall for a given rotational speed is one diameter. For the three fans installation, the results show a limit around 1.6 diameters as minimum distance between two fans. The recorded noise pressure levels were found almost independent of the fans distance.

Through the analysis of the geometric characteristics, aerodynamic and acoustic performance, conducted on a wide range of fans used in HAVC systems, it was found that there is a tendency in the industry to cover the engine located between the two fans in the machine for aesthetic reasons. A study of the effect on the aerodynamic performance of a fan for different covering solutions is also carried out. As is known, the engine causes a blocking effect in the aspirations closer to it, which implies asymmetric flow and other problems. The results show that any attempt to cover the engine area only intensifies this effect and consequently the problems associated with it.

sandrav@uniovi.es

SIMULTANEOUS MEASUREMENT OF AIR FLOW AND BLADE LOADING CONDITIONS IN AN AIR-COOLED STEAM CONDENSER FAN

Jacques MUIYSER / University of Stellenbosch (South Africa)
Danie N. J. ELS / University of Stellenbosch (South Africa)
Sybrand J. VAN DER SPUY / University of Stellenbosch (South Africa)

Axial flow fans used in large-scale air-cooled steam condensers (ACSCs) may operate under distorted inflow conditions. These conditions occur due to the prevailing wind conditions, the presence of buildings, and the location of the fan in the ACSC. Fans located on the periphery of the ACSC are affected the most due to their exposure to strong winds and the inner fans drawing in air past them. The distorted inflow causes cyclic blade loading conditions.

The purpose of the investigation is to simultaneously measure the inlet flow and the blade loading conditions of a single fan located on the periphery of a large-scale ACSC. The measurements were taken in real-time in order to compare any air flow variations with structural loads experienced by the fan system.

To measure the inlet flow a combination of ultrasonic and propeller anemometers along with thermocouples were used. One of each sensor was attached to six separate frames which were placed on the safety grid at the fan inlet. The anemometers were used to create a three-dimensional velocity vector for the air flow while the thermocouples measured the air temperature. These inlet flow measurements were coupled to measurements taken from eight propeller anemometers attached at the outlets of the heat exchanger bundles.

Blade loading was measured with strain gauges attached at the neck of the blade being monitored. Two sets of strain gauges were used to measure the bending in the flap-wise direction and the direction of rotation while another set was used to measure the torsion at the neck of the blade. The strain gauge data was transmitted to a base station which was synchronized with the flow measurements. Strain gauges were also attached to the fan's driveshaft to determine the load being transferred to the gearbox. These strain gauges also measured bending in two directions as well as torsion and were attached to the same wireless bridge amplifier.

To simplify the data capturing system all signals were converted to standard analogue voltages or currents. This simplification enabled the use of data capturing hardware with a large amount of channels, which in turn enabled easy synchronization of the multitude of sensor signals.

When installed, the system sufficiently captured data from the array of sensors and was able to perform autonomously for several hours at a time. During testing the ambient conditions were recorded using a weather mast located in the vicinity of the ACSC.

14903709@sun.ac.za

Testing an Axial Flow Fan Designed for Air-Cooled Steam Condenser Application

Sybrand J. VAN DER SPUY / Stellenbosch University (South Africa)
Theodore W. VON BACKSTRÖM / Stellenbosch University (South Africa)
Detlev G. KRÖGER / Stellenbosch University (South Africa)
Phillipe R.P. BRUNEAU / Sasol (South Africa)

Air-cooled heat exchangers (ACHEs) and steam condensers (ACSCs) offer a substantial benefit to the process and power generation industries, primarily by virtue of their sustainable use of cooling water. This technology is therefore particularly attractive to regions where cooling water is expensive or its availability is limited. An ACHE or ACSC consists of a rectangular array of axial flow fans that supply cooling air to a bank of heat exchangers. Based on their location within the array, for instance around the perimeter, the fans may be subjected to distorted inlet conditions. The distorted inlet conditions may be exacerbated by the presence of atmospheric winds.

A set of 1.5 m diameter axial flow fans, referred to as the B1- and B2-fans, were designed to address specific areas of concern associated with axial flow fans in an ACSC. These are the distorted inlet conditions that occur upstream of the fans and the occurrence of reverse flow at the fan hub (Venter, 1990). The B1- and B2-fans were designed for a specific design point while attempting to improve the efficiency of the fan. The format of the fans was constrained for practical reasons to be an 8-bladed rotor without stationary blade rows. This publication details the experimental evaluation of the B1- and B2-fans in comparison to an existing fan design. The design procedure followed in the development of the B1- and B2-fans is addressed in a separate publication.

Individual tests for the B1- and B2-fans were performed on a BS848 Type A fan test facility. The tests were performed for an operating domain that straddles the design point specified previously. The experimental results were compared to results obtained for a comparative industrial fan, referred to as the V-fan. The results showed that the B1- and B2-fans exhibit a higher fan static efficiency at the design point, when compared to the V-fan.

Numerical simulations performed with the B-fan showed that it may exhibit a high volumetric effectiveness when used in a multiple fan installation (Bredell et al., 2005). 630 mm diameter models of the B2-fan and V-fan (referred to as the B-fan and N-fan) were therefore manufactured and tested in the perimeter fan position in a multiple fan test facility (Conradie, 2010). The facility consisted of three fans, in parallel, extracting air from a common inlet chamber. One of the three fans resembled a perimeter fan, while the format of the two inner fans was left unchanged. The floor height of the inlet chamber was adjusted to vary the level of inlet distortion experienced by the perimeter fan.

The multiple fan results showed that the B-fan and N-fan exhibit a similar volumetric effectiveness at large floor heights. At the lowest floor height the volumetric effectiveness of the N-fan is however considerably lower than that of the B-fan. When considering the fan shaft power consumption in relation to the volumetric effectiveness of the B- and N-fans, the benefit of the higher efficiency of the B-fan is apparent.

sjvdspuy@sun.ac.za

SIMPLIFIED THEORY OF A RISK OF THERMAL CATASTROPHE IN AIR-COOLED ELECTRONIC CIRCUITS

Roman VINOKUR / Resmed Motor Technologies (United States of America)

The goal is to evidently illustrate a risk of thermal catastrophe in electronic circuits cooled by air. As known, the heat produced in an electric current, if not dissipated, may notably elevate the temperature of the circuit and damage its components. In circuits with a negative thermal coefficient of resistance (which is typical for the elements made of carbon and semiconductors), such a transient process can follow a pattern of thermal catastrophe. The theoretical model includes a simple resistor under constant voltage; the dominant mechanism of heat dissipation is a forced thermal convection in air; the temperature coefficient of resistance can be negative or positive. Despite a relative simplicity of the model, the author has never come across such a study in the literature for case of negative temperature coefficient of resistance.

The main results are the following:

(1) if the temperature coefficient of resistance is positive, a stable equilibrium can be achieved at some temperature;

(2) if the temperature coefficient of resistance is negative but the air cooling is efficient, the thermal equilibrium may formally exist at two temperatures, only one of such equilibrium states being stable;

(3) if the temperature coefficient of resistance is negative and the air cooling is relatively weak, a thermal equilibrium is not feasible at all and the temperature goes up unlimitedly for this mathematical model. It is noteworthy that such an effect might have happened in Edison's experiments on his first electric bulbs because the filaments were made of carbonized materials. Simple equations and plots have been derived to describe the key physical phenomena qualitatively. Certainly, the real-life conditions are more complex but the theory clearly describes the main trends and can help practical specialists and engineering students better understand the importance of adequate air cooling systems in computers and other electronic devices, in particular those utilized for real-time control.

romanv@resmed.com

On the Role of Leading-Edge Bumps in the Control of Stall On-Set in Axial Fan Blades

Alessandro CORSINI / Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)
Giovanni DELIBRA / Dept. Mechanical and Aerospace Engineering, Sapienza University of Rome (Italy)
Geoff SHEARD / Fläkt Woods Limited (United Kingdom)

Biologists have associated the capability of the humpback whale to execute very sharp rolls and loops under water with the presence of bumps on the leading edge of their flukes that work as a stall-control system. Taking a lead from the humpback whale flukes, this paper reports on a three-dimensional numerical study of sinusoidal leading edges on cambered airfoil profiles. The authors computed the turbulent flow around the cambered airfoils at different angles of attack with the open source solver OpenFOAM and an isotropic eddy viscosity model integrated to the wall. The reported research focused on the effects of the modified leading edge in terms of lift-to-drag performance. The research was primarily concerned with the elucidation of the fluid flow mechanisms induced by the bumps and the impact of those mechanisms on airfoil performance.

corsini@dma.ing.uniroma1.it

3D UNSTEADY CFD SIMULATION OF THE UNSTEADY FLOW IN A CENTRIFUGAL FAN

Yang YANG / Clausthal University of Technology (Germany)
Andreas LUCIUS / Clausthal University of Technology (Germany)
Gunther BRENNER / Clausthal University of Technology (Germany)

The purpose of this paper is to study the pressure fluctuation in a high performance centrifugal fan. Using Computational Fluid Dynamics (CFD), the three – dimensional unsteady flow field in the whole domain of impeller and volute casing is solved. Since these fluctuation are causing severe dynamic loads their assessment is crucial during the design process in view of life time prognoses. The Scale Adaptive Simulation (SAS) is used to model the turbulence in Navier – Stokes equation. The unsteadiness of the flow is induced by the interaction of impeller – volute and the complex geometry of the impeller. This paper focuses on the pressure fluctuation depending on these two phenomena. The calculations of the impeller with traditional spiral volute and combination of vaneless diffuser and collection volute are carried out at design point. Because of the rotating motion the volute and impeller domains are analyzed separately. The results show a periodical unsteadiness of the pressure in both cases. In the pressure field, the blade passing frequency and its multiples dominate. Due to the interaction between impeller and volute, the pressure fluctuation is strong at the impeller outlet and in the vicinity of the tongue considering the volute domain. With the application of the vaneless diffuser and the collection volute the distance between the impeller outlet and the tongue is increased. In addition, the diffuser effect is decreased because of the deceleration of the flow. Compared to the traditional spiral volute, the pressure fluctuation is considerably reduced through the application of vaneless diffuser and collection volute. The pressure distribution in the impeller is analyzed in rotational coordinates. In both cases the pressure fluctuation remains at the same level. As the flow changes its direction from axial to radial in the impeller, the pressure fluctuation reaches its maximum, which is almost three times higher than the one in the volute. The paper underlines the importance of numerical flow simulation and accurate turbulence models in understanding the origin of flow instabilities and qualifying their impact on the compounding dynamic loads.

yang.yang@tu-clausthal.de

INVESTIGATION OF UNSTEADY FLOWS IN A CENTRIFUGAL FAN USING HIGH-SPEED PIV AND NUMERICAL SIMULATIONS

Andreas LUCIUS / Clausthal University of Technology (Germany)
Gunther BRENNER / Clausthal University of Technology (Germany)
Andreas LEHWALD / Otto-von-Guericke University (Germany)
Dominique THEVENIN / Otto-von-Guericke University (Germany)

In the design of centrifugal fans the trend goes towards increasing circumferential speed with higher loads. In addition to the centrifugal forces, fluctuating fluid forces become more and more important. A major source of unsteadiness is the rotor stator interaction between the impeller and the volute casing. In this case the excitation frequency is well known in advance and can be considered during the design process. Especially in part load flow instabilities with unknown frequencies like rotating stall may arise, which can lead to severe dynamic loads and structural vibration. Due to the increased availability of computational power, unsteady flow simulations of the entire fan including inlet ducting, impeller and volute casing become feasible on high performance clusters. With a proper prediction of unsteady fluid forces, the designer is able to account for possible flow induced vibrations. In order to reach this aim, adequate numerical modeling is required. This includes especially sufficient grid resolution and accurate turbulence modeling. Hybrid LES RANS modeling is regarded as a promising compromise, allowing the resolution of large scale turbulent motion, while avoiding a full LES resolution of attached turbulent boundary layers. Since these advanced models should not be applied as black box, experimental results are necessary for validation.

In order to provide high quality experimental results for validation of CFD simulations, High-Speed PIV investigations of the flow field in the impeller of a centrifugal fan have been conducted. The impeller and parts of the casing were made from plexiglass for optical access. For this reason the rotational speed is limited to 2500 rpm. The PIV measurements cover several rotational speeds and operating points in the whole operating range of the fan. All three components of the velocity were measured in the center plane of the rotor using stereoscopic PIV.

Numerical simulations were done covering several approaches at selected operating points. Steady state simulations using the standard SST turbulence model were applied near design point. Transient simulations were conducted in design point and at deep part load. Beside the traditional SST model, the Scale Adaptive Simulation (SAS) approach was applied. This model is able to resolve turbulent motion with sufficient grid resolution and time step, with reduced grid dependence in comparison to the more common Detached Eddy Simulation (DES) approach. An extensive comparison of the different approaches with the measurement data will be presented at the conference.

andreas.lucius@tu-clausthal.de

INVESTIGATION ON THE UNSTEADY AERODYNAMICS OF AN INDUSTIRAL FAN

Domenico BORELLO / Sapienza University (Italy)
Alessandro CORSINI / Sapienza University (Italy)
Franco RISPOLI / Sapienza University (Italy)
Geoff SHEARD / Fläkt Woods Ltd (United Kingdom)

There are controversial requirements involved in developing numerical methodologies in order to compute the flow in industrial fans. The full resolution of turbulence spectrum in such high Reynolds number flow configurations entails unreasonably expensive computational costs. The authors applied the study to a large unidirectional axial flow fan unit for tunnel ventilation to operate in the forward direction under ambient conditions. This delivered cooling air to the tunnel under routine operation, or hot gases at 400°C under emergency conditions in the event of a tunnel fire. The simulations were carried out using the open source code OpenFOAM, within which they implemented a (very) large eddy simulation (VLES) based on one-equation SGS model to solve a transport equation for the modelled (sub-grid) turbulent kinetic energy. This sub-grid turbulence model improvement is a remedial strategy in VLES of high-Reynolds number industrial flows which are able to tackle the turbulence spectrum's well-known insufficient resolution. The VLES of the industrial fan permits to detect the unsteady topology of the rotor flow. This paper explores the main secondary flow phenomena's evolution and speculates on its influence on the actual load capability when operating at peak-pressure condition.

domenico.borello@uniroma1.it

TONAL NOISE OF AN ISOLATED AXIAL FAN ROTOR DUE TO INHOMOGENEOUS COHERENT FLOW STRUCTURES AT THE INTAKE

Michael STURM / University of Siegen (Germany)
Thomas CAROLUS / University of Siegen (Germany)

The overall acoustic performance of a fan is sometimes dominated by the tonal components, especially at blade passing frequency (BPF) and higher harmonics. In the case of an isolated fan rotor with a perfectly symmetric inlet nozzle, running at low characteristic circumferential Mach number, no BPF noise should exist, while measurements show just the opposite. Many authors have shown that the tonal noise at BPF is closely linked to ingested flow inhomogeneities. However, there is no consensus about what causes those incoming inhomogeneous flow structures and the subsequent BPF noise in the case of an isolated rotor.

An isolated fan is investigated on a standardized measurement test rig in an anechoic room. Supporting struts are positioned one rotor diameter downstream, the inlet nozzle is perfectly round with a large 1/4 rotor diameter radius and no obstructions are present. Hence, up- and downstream disturbances are thought to be minimal. The blades are instrumented with several flush mounted miniature pressure transducers on the suction and pressure side as well as at the tip. Hot-wire measurements are made in a plane 1/2 rotor diameters upstream of the rotor plane to characterize the incoming flow. The sound pressure is measured by microphones at different positions around the inlet and an in-duct microphone. A state-of-the-art hemispherical inflow control device (ICD), as used for aircraft engine tests, is employed to homogenize the incoming flow. The tones at BPF and higher harmonics are clearly reduced with ICD but they do not disappear completely.

A correlation technique is applied to investigate the flow characteristics as measured by the flush mounted pressure transducers. Nearly stationary coherent structures are detected. A flow visualization at the intake reveals stretched helical vortex structures exist in the intake which may act as a source of the tones.

michael.sturm@uni-siegen.de

AN INVESTIGATION OF THE EFFECT OF UNEVEN BLADE SPACING ON THE TONAL NOISE GENERATED BY A MIXED FLOW FAN

Ludovic DESVARD / Dyson Ltd (United Kingdom)
Jeremy HURAULT / Dyson Ltd (United Kingdom)
Muhammad Affendi Bin MOHAMED ZAMZAM / Dyson Ltd (United Kingdom)
Ash SYMES / Dyson Ltd (United Kingdom)

The acoustic signature of a desk fan is one of the keys to its market success. In a Dyson Air Multiplier fan, the internal air flow is generated by a mixed-flow impeller with a low blade tip Mach number. The impeller produces an audible tone, known as the blade passing frequency tone. This sharp tone is inherent to all fans and compressor units and is problematic to desk fans since it could be perceived by the user against the fan broadband noise. The two main factors influencing the blade passing frequency are the rotational speed of the impeller and the number of blades. The approach discussed in this paper was to develop a modulated (uneven) blade spacing mixed-flow impeller (driven by a DC brushless motor) in order to spread the associated energy from a single tone to several frequencies. First, a theoretical analytical tool was developed to define a suitable uneven blade configuration and the predicted acoustic signatures with associated relative magnitudes were calculated. The unevenly spaced blade impeller was then prototyped to measure aerodynamic and acoustic performance against a symmetrically spaced blade impeller using an ISO5136 test rig. Also presented in the paper are the results of a perceptive test conducted to establish a sound quality model for desk fans. The sound quality model quantifies the level of subjective preference for desk fan sound and was used to assess the acoustic performance of both the uneven and symmetrical blade spacing impeller designs. Results show that the uneven blade spacing impeller does not improve the subjective preference compared to the symmetrically spaced blade impeller.

Keywords: Tonal noise, Modulated blade spacing, Mixed-flow impeller, Sound quality.

ludovic.desvard@dyson.com

INVESTIGATION INTO THE SOUND POWER LEVEL REDUCTION ACHIEVED USING ACOUSTIC JACKETS ON A CENTRIFUGAL FAN AND AN AXIAL FAN

Mark SIMMS / Ventac Group (Ireland)

A very common method to reduce the noise radiated from a fan is to wrap the impeller casing with an acoustic jacket. This project seeks to measure how much an acoustic jacket reduces the noise from a typical fan and whether there are any opportunities for improvement. As part of the testing, axial and centrifugal fan configurations were used. The acoustic jacket material that was used was made from vinyl faced acoustic quilting and a 5kgm-2 barrier layer.

The A-weighted sound power level was measured using sound intensity measurements across a measurement surface encompassing the impeller casing, motor and a section of the inlet and outlet duct work. The measurements were also broken down in to the third octave frequency bands. The sound intensity measurements allowed for the generation of noise maps of noise radiating from the source.

The jackets were found to perform well on the axial fan as it was straight forward to enclose the noise source but for the centrifugal fan (3kW direct drive) it became apparent that the cooling fan on the motor was a key source. An acoustic jacket was developed to extend across the motor but not impede its performance in cooling air. This resulted in a reduction of the sound power radiated from the source down to the level of the break out noise from the ducting.

The noise mapping from the sound intensity measurements provided data in a visual format which demonstrated the relative importance of the different elements of the noise source including; scroll casing, motor and connecting ductwork which was important in developing the product. The main aim of this project was to produce data on the sound power level reductions that can be achieved using acoustic jackets on typical fan set-ups that would be found in the field.

msimms@ventac.com

FAN NOISE CONTROL: CASE OF DATA CENTRES

Jean-Marie VERLHAC / CETIM (France)

In order to meet demand of ever-growing internet flows, data centres are multiplying in urban and rural areas. While energy cost of a Google search is already a care (A. Wissner Gross's work, PhD, Harvard University physicist), noise of web industry remains quite unknown.

Indeed, such impressive gatherings of computers form true industrial facilities. They can host up to several dozens of megawatts, mainly for the needs of ventilation (servers and cooling systems). These equipments generate noise levels that are comparable to those of mechanical industry.

Given the demanding regulatory framework (2002/49/CE European Directive, French regulation pour industrial facilities, etc), their noise impact is not to be neglected.

In light of various studies as a client assistant for noise control of these server farms, CETIM draws up the acoustic stakes that are specific to this field.

A risk management approach is brought up, at each step of the project: choice of site location, choice of cooling technologies and machinery, room design and development, facility operation…

This method takes into account this sector's strong demands:

- dynamism of the market, requiring short study and project deadlines
- profitable of room floor area, leaving few space for auxiliary equipment
- special requirements of key accounts concerning room design and fitting
- service continuity requirement, which means highest standards of operating safety, and making facility modification nearly impossible during operation

At each step, obstacles are pointed out, such as during regulation and standards interpretation, identification of noise exposure risk zones, reading of manufacturer data sheets, or in matching noise abatement solutions with the type of noise to be treated.

Case studies provide examples of noise abatement reached in collaboration with the contracting owner, project management and suppliers.

jean-marie.verlhac@cetim.fr

SUPPRESSION OF FAN NOISE OF HIGH CIRCUMFERENTIAL MODE USING OPTIMIZED DUCT WALL PROPERTIES

Lixi HUANG / Department of Mechanical Engineering, The University of Hong Kong (China)

Sound absorption by duct lining is most effective in the medium frequency range. The attenuation spectrum tails off at both low- and high-frequency extremes. The reduction of absorption rate towards the high frequency end is more difficult to deal with, especially when the source has complex circumferential and radial modes.

This work investigates, theoretically, the absorption of high frequency, high circumferential order rotor noise by a wall lining of finite axial length. The wave equation is solved in the frequency domain with the Chebyshev collocation method. The duct has a segment of lining with a typical length equal to the casing diameter. The scattered, out-going waves are propagated out perfectly using a non-local, Dirichlet-to-Neumann condition that covers all the modes resolved by the Gauss Lobatto grid. The incident wave simulates a typical concentrated source located near the circumference of the duct, which may be expanded into multiple radial modes. An axially distributed, locally reactive impedance is used for the duct-lining interface. An optimization scheme is used to maximize the transmission loss.

For a given frequency, it is shown that such optimization may yield significant extra noise reduction when compared with the baseline configuration of a uniform lining which is itself optimized for a frequency. Details of the acoustic field are analyzed to reveal the mechanism of the high absorption rate derived from the optimized liner.

lixi@hku.hk

THE INFLUENCE OF THE REYNOLDS NUMBER AND ROUGHNESS ON THE EFFICIENCY OF AXIAL AND CENTRIFUGAL FANS – A PHYSICALLY BASED SCALING METHOD

Peter PELZ / Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)
Stefan STONJEK / Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)
Berthold MATYSCHOK / Chair of Fluid System Technology, Technische Universität Darmstadt (Germany)

Acceptance tests on large fans to prove the performance (efficiency and total pressure rise) to the customer are expensive. Hence there is a need to reliably predict the performance of fans not only at the design point but also at part- and overload from measurements on down-scaled test fans. The commonly used scale-up formulas give satisfactorily results only near the design point, where inertia losses are small in comparison with friction losses. At part- and overload the inertia losses are dominant and the scale-up formulas used so far fail. This work develops a new method, which aims to fill this gap. Furthermore, the new method considers the influence of surface roughness and geometric variations on the performance. The validation of the new scale-up formula is performed with test data from two axial fans with a diameter of 1000 mm / 250 mm and two centrifugal fans with 2240 mm / 896 mm diameter. Except Reynolds number, Mach number and relative roughness the two axial fans and the two centrifugal fans are similar to each other.

The method, discussed in this work, consists of two steps: Initially efficiency is scaled. Efficiency scaling is derived analytically from the definition of the total efficiency. With the total differential and one simplification it can be derived, that the change of friction factor is inversely proportional to the change of efficiency.

The second step allows for calculating the shift in flow rate, which can typically be observed in most test data. Prediction of this shift bases on a physical model concerning the displacement thickness of the boundary layer on the blades in a fan.

Experimental results concerning the efficiency of the four fans at different rotational speeds (i. e. Reynolds numbers) are shown. The new correction method is validated and comparad to other methods. The predicted performance characteristics show a good agreement to test data.

pelz@fst.tu-darmstadt.de

EXPERIMENTS ON THE USE OF SYMMETRIZED DOT PATTERNS FOR IN-SERVICE STALL DETECTION IN INDUSTRIAL FANS

Stefano BIANCHI / Sapienza University (Italy)
Geoff SHEARD / Fläkt Woods (United Kingdom)
Alessandro CORSINI / Sapienza University (Italy)

Aerodynamic instability when throttling flow rate constrains the operational envelope of fans, blowers and compressors. To avoid instability, aerodynamicists must provide adequate stability (stall) margin to accommodate inlet distortions, degradation due to wear, throttle transients, and other factors that reduce fan, blower and compressor stability from the original design base-line.

Because aerodynamic stall, resulting in increased alternating aerodynamic loads, is a major potential cause of mechanical failure in axial fans stall-detection techniques have had wide application for many years.

The detection and analysis of the different forms of aerodynamic instability have been studied for several decades. Two main types of aerodynamic flow instability exist: (i) 'rotating stall' (in which regions of reversed flow occur locally); and (ii) 'surge' (which is characterised by periodic backflow over the entire annulus involving violent oscillations in the air flow which can result in mechanical failure such as fan blade breakage.

Prior methods have had drawbacks in their inability to enable a sufficiently rapid response to the onset of stall to avoid damage and their inability to sense the approach to stall. First, alert methods have been identified for individual test beds but reliable warnings of general validity require further research. Second, techniques for the detection of stall initiation based on experimental observation of pre-stall behaviours have sought to identify such behaviour as early as possible to enable an active control system to react and suppress the incipient stall. As a third factor, all the non-model based detection techniques rely on the use of on-board probes able to sense the unsteady pressure evolution in the vicinity of the blade rows, i.e. usually flush-mounted on the casing.

The paper describes a stall-detections criterion based on the use of symmetrized dot pattern (SDP) visual waveform analysis and the stall-warning methodology based on that analysis recently developed. The present programme of work is aimed at the verification of the SDP based early-stall warning technique in a test environment relevant of the in-service operation of a cooling fan unit.

The experimental study explores the capability of the SDP technique to detect the stall incipience and evolution in presence of low signal to noise ratios, i.e. noisy working environment. Moreover, the investigation presents a systematic analysis on the influence of the probe position with respect to the fan section. As such the SDP technique in combination with an acoustic measurement is able to create a visual pattern that can be used to detect stall in any locations, not just with the microphone over the blade itself.

bianstefano@gmail.com

INLET INSTALLATION EFFECTS ON DIFFERENT TYPES OF FANS AND DUCTWORK DESIGNS

Alain GUEDEL / CETIAT (France)
Bill CORY / William Cory Consultancy (United Kingdom)
Mark STEVENS / AMCA International (United States of America)

It is known that installing a ductwork fitting at the inlet of a fan can have a significant effect on the performance curve of the fan. Conversely, when the fitting is connected to the fan outlet the performance curve is not significantly modified but the pressure loss of the fitting itself may considerably increase due to the strong non-homogeneity of the flow velocity at the fitting entrance.

This paper presents an analysis of test results obtained by NEL and AMCA with the financial support of FETA and ASHRAE respectively in order to quantify the fan system effect due to various duct singularities or obstructions (bends of different shapes, duct contractions, bearings, plenum with lateral opening, wall close to the fan, …) at the inlet of axial or centrifugal fans of different types. Unlike the methodology described in AMCA Publication 201 the system effect is assessed here by calculating the drop in flow rate due to the presence of the inlet appurtenance on a system resistance curve. The pressure loss of the fitting itself is subtracted from the total fitting performance penalty in order to assess the actual system effect. An average of the relative flow drop on several system curves is then calculated, which quantifies the system effect of each inlet obstruction for a given fan.

The amount of system effect strongly depends on the fitting shape and type of fan with a relative flow reduction which may exceed 20 % in some extreme cases and is totally insignificant in other cases.

This approach has been extended to quantify the system effect on fan power and sound power levels due to inlet fittings. A few test data are presented to illustrate the analysis in this case.

alain.guedel@cetiat.fr

CFD ELECTRIC MOTOR EXTERNAL FAN SYSTEM COMPARISON AND VALIDATION

Cassiano A. CEZARIO / WEG Equipamentos Eletricos - Motores (Brazil)

This work presents a comparison between different approaches that can be used to evaluate the external fan system of an electric motor in a Computational Fluid Dynamics (CFD) code. The entire process of comparison was supported by the experimental data, which was obtained from a special device. This device was designed and manufactured in order to consider a number of geometric simplifications aiming the validation of a CFD model. The geometry of the experimental device was transposed to commercial Computer Aided Design (CAD) software with insignificant adjusts, only when it was extremely necessary, to avoid numerical problems in CFD software. With this approach it was possible to concentrate the efforts in numerical problems, avoiding questions like, were the differences between numerical and experimental data originated by numerical error or geometrical considerations?

From numerical point of view the impact of turbulence model is a critical point in CFD simulations, so the turbulence model Shear Stress Turbulence (SST) and k-ε were confronted, both based on Reynolds Average Navier-Stokes (RANS). Especial attention was dedicated to the variation of the y+, i.e., quality of the mesh near the wall. Additionally, the problems of numerical convergence in steady-state regimes were discussed in this work. The numerical results were confronted with air velocity and fan power consumption, both experimentally obtained, and showed good agreement in external flow.

For all simulations, CAD software adopted was SolidWorks 2009 SP2.1 and the CFD software was ANSYS-CFX 13.0 SP2. The mesh dimensions exceed 5.000.000 of nodes and the processing was made in two HP Z800 Workstations with two Xeon X5690 (six-core) processors and 24 GB memory, each.

cassiano@weg.net

NUMERICAL INVESTIGATION OF AXIAL FANS IN SERIAL CONNECTION

Gabriel AXTMANN / University od Stuttgart (Germany)
Michael SCHMITZ / ebm-papst St. Georgen GmbH & Co. KG (Germany)
Wolfgang LAUFER / ebm-papst St. Georgen GmbH & Co. KG (Germany)

Compact fans are often used for electronic cooling for example in telecom base stations or large computer servers. In many cases two axial fans are operated in two-stage configurations, sometimes six or more of those sets are used per application. Most of the time, these fans are operated at lower part load only in case of failure of one fan the remaining fan will speed-up to avoid a local pressure drop or flow short-cut in the application. Another reason for multistage arrangement of fans is to get an overall performance curve towards more pressure. Typically, two identical standard fans are used for this purpose, sometimes with flow straighteners in between. For this study, a two-stage set-up of compact fans with impeller diameter of 75mm is investigated by 3D numerical methods. Steady and unsteady calculations of different arrangements are investigated and the results are compared to experiments. It can be shown, that a proper arrangement of two axial fans will increase both static pressure rise and flow rate. In addition to the aerodynamic performance, the acoustic behavior of the different set-ups is presented. It is shown, that the arrangement itself and the additional components have significant impact on the acoustic behavior with respect to sound power and sound quality.

gabriel.axtmann@de.ebmpapst.com

CFD Techniques Applied to Axial Fans Design of Electric Motors

Samuel Santos BORGES / Weg Equipamentos Eletricos - Motores (Brazil)

The objective of this work is to present axial fan applications in electric motors. Electric motors are widely used by humans to convert electric energy into mechanical energy. This converting process is characterized by generating losses, which are transformed into heat. To ensure the quality, safety, performance and/or efficiency of the motor this generated heat must be removed, therefore, most of the motors comprise air cooling systems. Among the components of the cooling systems, the fans paramount. The fans can be used externally, internally or both in an electric motor, these configurations depend on the motor characteristics and its applications.

A correct fan design is very important for the temperature, losses and noise of the motor to be kept below the limits imposed by standards and/or customers. Therefore, the engineer or designer must apply the concepts of turbomachines and electric machines to properly design cooling systems.

The fans can be comprised of axial or radial rotors. In this approach, the axial rotors will be focused. Advantages and disadvantages, design fan concepts, design using computational fluid dynamics (CFD) and cases studies, will be presented.

Many design details will be considered, such as: applications and assembling, typical curves, similarity laws using CFD, CFD validation, fan design and fans troubleshooting.

samuelb@weg.net

NUMERICAL AND EXPERIMENTAL INVESTIGATIONS OF SINGLE-FLOW AND BYPASS-FLOW FANS

Victor MILESHIN / Central Institute of Aviation Motors (CIAM) (Russia)
Igor OREKHOV / Central Institute of Aviation Motors (CIAM) (Russia)
Sergey PANKOV / Central Institute of Aviation Motors (CIAM) (Russia)

During design procedure it is necessary to take into account all characteristic features of fan operation in real-life environment. Defining of bypass ratio influence on local flow parameters and integral performances of fan is time-consuming task. This work highlights a method of computing bypass fan performances on the basis of through-flow 3D viscous flow computation within RANS based on S.K. Godunov's modified implicit scheme. The value of the specified static pressure at the outlets of core and bypass ducts is used as a boundary condition in this work. The strategy in finding solution is similar to the method used in experimental studies of bypass fans when using two independent throttles in each duct.

Numerical predictions of integral performances and steady flow inside the blade-to-blade channel of bypass fans were performed for the reference fan and two counter rotating fan models, investigated in frame of European project VITAL. This paper represents results of computed and experimental data for two counter rotating fan models - CRTF1 and CRTF2, investigated in frame of European project VITAL. Good correspondence of computed and experimental data is observed. The abovementioned solver allows performing calculations of several fan stages in the flow path including the flow split to the core and bypass ducts. In bypass ducts the guide vane OGV, struts and pylons may be taken into consideration. In core ducts the guide vanes IGV, AGV, transition ducts, struts and several stages of booster, for example, or LPC, HPC may also be taken into account.

Numerical and experimental investigation of fan stage S179-1 with the rotor wheel manufactured according to “blisk” technology was performed. Fan stage S179-1 is the scaled model – prototype of bypass fan for advanced turbofan engine. Also numerical and experimental study of bypass fan stage S179-2 was performed, which is the scaled model of the whole bypass fan including four stages booster. For both cases computations showed good correspondence in comparison of numerical and experimental data.

mileshin@ciam.ru

A NUMERICAL ANALYSIS ON THE AERODYNAMIC NOISE OF CROSS-FLOW FAN BY USING A 2-D URANS SIMULATION AND ACOUSTIC ANALOGY

Myungsung LEE / Sogang University (South Korea)
Seongwon KANG / Sogang University (South Korea)
Nahmkeon HUR / Sogang University (South Korea)
Jeongtaek PARK / LG Electronics (South Korea)

The aerodynamic noise of the cross-flow fan (CFF) in an air-conditioner (AC) was investigated by using the computational fluid dynamics (CFD) and acoustic analogy. Two-dimensional unsteady flow field of CFF was obtained by solving the compressible Reynolds-Averaged Navier-Stokes (RANS) equation in order to capture the acoustic wave generated from the CFF. The moving mesh technique was adopted at the arbitrary sliding interface between the region rotating with impeller and rest of the region including all stationary parts of AC such as stabilizer, rear guider, etc. The propagated acoustic pressure from the CFF to a microphone was predicted by using the Ffowcs Williams-Hawkings (FW-H) acoustic analogy equation. In the present study, various operating conditions of the CFF were considered in the range of 877-1130 rpm. The results of CFD simulation showed the typical flow field of the CFF such as eccentric vortex which was observed in the vicinity of the stabilizer. Since the flow rate of the CFF is strongly dependent upon the location of the vortex core, the accurate prediction of the location of eccentric vortex is very important. To validate the present numerical analysis, the numerical results were compared with available experimental data such as the sound pressure level (SPL) spectrum and overall SPL. The tonal noise level at the blade passing frequency (BPF) was well predicted in the present study. On the other hand, the prediction accuracy of broadband noise in high frequency was relatively poor. This is because the broadband noise is generated due to the random flow characteristics like turbulence which RANS cannot consider well. The result of the present study, however, can be used for the optimal design of the AC components to reduce the peak sound pressure level which is associated with the tonal noise at BPF.

cfdlove@sogang.ac.kr

INFLUENCE OF BLADE NUMBER ON AERODYNAMIC NOISE OF PROPELLER FANS FOR OUTDOOR UNIT OF AIR-CONDITIONER

Taku IWASE / Hitachi, Ltd. Hitachi Research Laboratory (Japan)
Tetsushi KISHITANI / Hitachi Appliances, Inc. (Japan)
Masato FURUKAWA / Department of Mechanical Engineering, Kyushu University (Japan)

In this study, flow fields in half ducted propeller fans for outdoor unit of air-conditioner have been calculated by a finite element method-based Large Eddy Simulation (LES) with the aim of investigation for influence of blade number on aerodynamic noise. 2-blade and 4-blade propeller fans were intended. Noise level of the 2-blade propeller fan was smaller than the 4-blade by 5.1dB experimentally. Dynamic Smagorinsky model was used as sub-grid scale model. Computational grid consisted of three parts that were inlet part, propeller part, and outlet part. The grid is composed of hexahedral elements. Numbers of grid elements for inlet and outlet parts were approximately 2 million and 5.5 million elements, respectively. Numbers of grid elements of fan neighborhood part for the 2-blade and 4-blade propeller fan were approximately 5.5 million and 10.6 million elements, respectively. Aerodynamic noise was calculated by Curle's equation based on pressure fluctuation calculated by LES. The calculated static pressure rise and shaft power showed reasonable agreement with the experimental results. The calculated time-averaged velocity distributions downstream of the blades were also compared with experimental results by hotwire anemometry. The velocity distributions showed reasonable agreement between the calculated and experimental velocity profiles. But the calculated velocity fluctuations downstream of the blades were overestimated the experimental results. Calculated noise level of the 2-blade propeller fan were smaller than that of the 4-blade. Calculated difference of overall noise level was 5.7 dB. Moreover, we confirmed that the tip vortex and leading edge separation vortex had a great influence on half-ducted propeller fans. The tip vortex trajectory and the blade pitch of the 2-blade propeller fan were longer than those of the 4-blade propeller fan. These suppressed the interaction between the tip vortex, the ring, and the adjacent blade. The 2-blade propeller fan was therefore more silent than the 4-blade propeller fan.

taku.iwase.mf@hitachi.com

Optimization of Axial Fans with Highly Swept Blades with Respect to Losses and Noise Reduction

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

The main objectives in axial fan design are high efficiency and low sound emission over a wide operating range. A successful design strategy with respect to noise is blade sweep. How-ever, blade sweep also increases losses due to secondary flow effects and thus decreases effi-ciency. The work presented deals with a redesign of a pre-existing highly swept axial fan con-sidering both, sound emission and efficiency. The old design is exclusively based on a quasi 2D blade element method whereas the new design strategy addresses 3D flow effects in more detail. Secondary flows were brought into special focus and examined by means of steady state CFD simulations and an optimization method.

Firstly, the magnitude and position of maximum camber of the utilized NACA profile were varied along the blade span. Secondly, a simple kind of axisymmetric hub contouring was applied to address losses linked with the near-wall circumferential pressure gradient. Geome-trical parameters were defined and the aerodynamic optimum was approached using numerous runs of RANS simulations and the Nelder-Mead optimization method. As target function the total-to-total efficiency at the design point has been selected. An important constraint in the optimization cycle was constant chord length in spanwise direction ensuring equal sweep at leading and trailing edge whereas in the old design the superposition of sweep angles speci-fied at the blade stacking line and variable chord length lead to sweep reductions either at the leading or trailing edge.

It becomes evident that both, a wave shaped hub contour along the blade channel and a new distribution of magnitude and position of maximum camber can manipulate unfavorable pres-sure gradients and hence reduce near wall secondary flow losses. The latter design feature means turning away from predefined spanwise load distributions (e.g. free vortex) and shifts more load towards the outer blade part. The angle of attack was not optimized but is still se-lected according to the 2D airfoil polars. Eventually the characteristic curves of the optimized design were predicted via RANS and experimentally validated. The test results showed good agreement with the simulations. The aerodynamic benefit of the new design was a considera-ble extension of operating range and a moderate increase in efficiency near design point. Moreover, acoustic investigations in a semi-anechoic chamber showed a considerable reduc-tion in sound power over the complete operating range.

konrad.bamberger@uni-siegen.de

STATIONARY AND MOVING SURFACE FFOWCS WILLIAMS AND HAWKINGS COMPUTATIONS OF AN ISOLATED RADIAL IMPELLER

Till HEINEMANN / University Erlangen-Nuremberg (Germany)
Christoph SCHEIT / University Erlangen-Nuremberg (Germany)
Matthias SPRINGER / University Erlangen-Nuremberg (Germany)
Stefan MÜLLER / University Erlangen-Nuremberg (Germany)
Stefan BECKER / Institute of Process Machinery and Systems Engineering,
University Erlangen-Nuremberg (Germany)

Numerical methods based on Lighthill's acoustic analogy can help to predict the noise emission of technical products such as radial fan impellers. The surface integral method of Ffowcs Williams and Hawkings (FW-H) uses the flow field data from a transient fluid computation as input. Pressure, velocity, and density distributions on an enclosed surface around the solid body represent all acoustic sources inside it. From this surface, the acoustic signals can be computed at observer points with distance-independent precision.

This work introduces two different FW-H formulations based on the flow field computed with a compressible unsteady Reynolds averaged Navier Stokes (URANS) simulation. Both FW-H formulations employ advanced time algorithms and comply with Farassat's Formulation 1, i.e. time differentiation is performed after surface integration.

The first algorithm bases on a stationary surface formulation. It uses the pressure, velocity, and density distributions on a stationary surface around the rotating solid body as input data. Advanced time interpolation and differentiation are optimized with respect to memory usage.

The second FW-H algorithm employs a moving surface formulation with input data from a co-rotating wrapping surface of constant shape on the solid body. The correctness of the code is proven with an analytical test case for different time and grid resolutions. Its technical application is demonstrated with an isolated radial fan impeller, with maximum blade tip velocities of Mach 0.12.

hei@ipat.uni-erlangen.de

NUMERICAL METHOD FOR 3D COMPUTATION OF TURBOMACHINERY TONE NOISE

Anton ROSSIKHIN / Central Institute of Aviation Motors (CIAM) (Russia)
Igor BRAILKO / Central Institute of Aviation Motors (CIAM) (Russia)
Sergey PANKOV / Central Institute of Aviation Motors (CIAM) (Russia)
Victor MILESHIN / Central Institute of Aviation Motors (CIAM) (Russia)

The numerical method for 3D calculation of the turbomachinery tone noise generation, propagation and radiation in the near and far fields taking into account the interaction between rows is developed in CIAM. The method is implemented in 3DAS (3 Dimensional Acoustics Solver) CIAM domestic solver. It was used for calculations of acoustic characteristics of high bypass fans of turbofan engines (single rotating and counter-rotating), counter-rotating propellers and last stages of low pressure turbines.

The method under consideration is based on the solution of three-dimensional Euler equations for disturbances (linear or nonlinear) in the reference frame of blade rows. Calculation consists of two stages. The first stage – computation of the mean steady flow field in turbomachine by Reynolds-Averaged Navier-Stokes equations, semi-empirical model of turbulence, and "mixing-plane" interfaces between blade rows. The second stage is the unsteady inviscid calculation of disturbances over the mean flow field, taking into account interaction between rows. This interaction is provided by so called “sliding grids” interface, which transfer mean flow disturbances, and unsteady disturbances, from one row to another. The method allows us to consider potential, wake and vortex interactions. The main features of the solver are:

- The fourth order DRP scheme (Dispersion Relation Preserving Scheme) is used for spatial approximation.
- Second order, four stages LDDRK scheme (Low Dissipation and Dispersion Runge-Kutta Scheme) is used for time derivative approximation.
- Calculation can be performed either in time or in frequency domain.

Usage of CAA methods allows us to perform calculations with less grid points per wavelength, and therefore to use relatively coarse grids.

For the far-field radiation calculation the Ffowcs Williams method with a penetrable data surface is used.

Application of our method for the calculation of tone noise of a model ducted counter-rotating fan at approach flight conditions is presented. The results of the computation are compared with the results of experiment.

rossikhin@ciam.ru

EVALUATION OF SQUIRREL-CAGE FANS FOR HVAC APPLICATIONS IN PUBLIC TRANSPORT: KEY PARAMETERS AND DESIGN GUIDELINES

Sandra VELARDE-SUAREZ / Universidad de Oviedo (Spain)
F. Israel GUERRAS-COLON / Universidad de Oviedo (Spain)
Rafael BALLESTEROS-TAJADURA / Universidad del Oviedo (Spain)
Jose GONZALEZ / Universidad de Oviedo (Spain)
Katia M. ARGÜELLES DIAZ / Universidad de Oviedo (Spain)
Jesus M. FERNANDEZ ORO / Universidad de Oviedo (Spain)
Carlos SANTOLARIA MORROS / Universidad de Oviedo (Spain)

Small centrifugal and axial fans are used in automobile applications, as it is the case for HVAC systems for public transport. In particular, a specific type of centrifugal fans named "squirrel-cage" is broadly used in these systems. Their main geometrical characteristics are a large number of short chord forward-curved blades and a rotor exit-to-inlet area ratio unusually large. Due to its size and relative high specific speed, these fans are used in applications with requirements of compact size, high flow rate and low cost. Typically, these fans are used to work at high rotation speed (over 4000 rpm) and variable operation conditions, even at extreme off-design points. Flow instabilities, low efficiency and high levels of noise and vibrations appear in these machines due to these working characteristics and to the need of low fabrication costs.

In this work, a morphological and functional characterization of several ventilation equipments used in commercial HVAC systems for public transport, is carried out. They all use an assembly of two squirrel-cage fans with drive motor incorporated.

First, a detailed description of geometric and construction parameters of the available equipments is made: diameters, thicknesses, drive system, building materials, etc. Following, the functional characterization of the machines is carried out, paying attention to aerodynamic, acoustic and energetic aspects: pressure, flow-rate, power consumption, total efficiency, sound power, etc.

All the information collected is expressed in terms of dimensionless parameters, making possible the comparison between the fans, independently of their size and drive speed. The specific speed ns ranges from 1.5 to 2.5 while the specific diameter ds shows values from 1 to 1.4. In general, performance curves with unstable shape have been obtained, with significant pressure and efficiency drop in the medium flow rate zone, coupled with increases in power consumption in this area. Total efficiencies are low in all the cases, in some cases staying in a poor 30% and not exceeding in any case 45%. In terms of acoustic generation, the results have been extremely diverse, with significant differences among the fans, but also in a particular machine when the operating point is varied.

The analysis of the obtained results is applied to select the best alternative among those analyzed, focusing on energy efficiency and minimum noise generation criteria. In this way, the most important design criteria are established in order to get ventilation systems more efficient and less noisy.

sandrav@uniovi.es

A Design of Experiment for Evaluating Installation Effects and the Influence of Blade Loading on the Aeroacoustics of an Automotive Engine Cooling Fan

Elias TANNOURY / Valeo Thermal Systems (France)
Bruno DEMORY / Valeo Thermal Systems (France)
Manuel HENNER / Valeo Thermal Systems (France)
Pierre-Alain BONNET / Valeo Thermal Systems (France)
Patrice CAULE / Technofan (France)
Yoann CRÉTEUR / Université de Poitiers (France)

Aeroacoustic performances of cooling fans in automotive industry have become a quality factor. However, tighter packaging and the quest for higher performance have made it more difficult to create quieter fans.

A specific design of experiment (DOE) has been carried out to assess the influence of various, commonly encountered geometrical parameters on the aeroacoustics of such cooling fans. It provided the necessary data to help find the best compromise between aerodynamics and aeroacoustics for a given packaging.

The work was conducted in two phases, the first one being a numerical study on blade loading, and the second one being an experimental study on the fan's interaction with its support:

- 3 fans delivering the same pressure rise at the target operating point were designed by the means of CFD. The first blade is loaded mostly near the bottom, whereas the second has greater load near the tip, and the third is designed to operate at lower rotational speed.
- 17 supports were designed according to a Nearly Orthogonal Latin Hypercube (NOLH) design of experiment based on one physical parameter (flow rate) and 6 geometrical ones. They include the rotor-strut distance, the strut's sweep angle, aspect ratio and cross-sectional area, the tip clearance and the rotating ring's immersion in the shroud.

The fans and the supports were manufactured and tested in an anechoic chamber equipped with a flow-rate-control rig. Each of the fans was tested with all the supports at the target operating point and several other flow rates. In total, 3 designs of experiment were carried out, each corresponding to a specific type of blade loading. The collected data was then used to create response surface models for efficiency and sound power.

Maximum efficiency, minimum noise, and a compromise between both configurations were then sought via the response surface models integrated into optimization loops. The results show that loading the blade near the tip makes it less sensitive to installation effects. Furthermore, maximum efficiency and minimum noise do not occur for the same combination of parameters. Future work should include a validation of the trends obtained by optimization.

elias.tannoury@valeo.com

Design and Aerodynamic Performance of High Pressure Axial Flow Fan

Vaclav CYRUS / AHT Energetika Ltd (Czech Republic)
Jan CYRUS / AHT Energetika Ltd (Czech Republic)
Pavel PANEK / AHT Energetika Ltd (Czech Republic)

Modern coal fired power stations fitted with desulphurization equipments require axial flow fans with high pressure coefficient and flow rate. The fans consist of one or two axial stages and inlet suction chamber and exhaust diffuser. A proposed paper will be devoted to design and experimental and theoretical investigations of aerodynamic performance of stage blading and complete fan for design rotor blades stagger angles. Flow simulations results were obtained by means of Numeca commercial code.

Fan stage with hub/tip ratio of 0,6 has design pressure and flow coefficients values: 0,83 and 0,60, respectively. A geometry of stage blade rows was determined by the application of a code for axial compressor stage design using streamline curvature method for axisymmetric flow solution. Rotor and stator blade elements have relatively high aerodynamic loading. It is expressed by diffusion factor originally devised by Lieblein. The maximum values of rotor and stator rows diffusion factor were 0,57 and 0,54.

Inlet chamber flow area has rectangular shape with sides ratio of 1,28. The inlet velocity vector is perpendicular to machine axis. Three variants of chamber struts were considered. Area ratio of exhaust diffuser is 1,87.

In the first step of aerodynamic research the fan stage was measured on test rig with external diameter of 600 mm. Obtained experimental aerodynamic performance proved that stage design targets were fulfilled. Maximum stage efficiency value was found :90,1% ±1.2%. Measured and computed aerodynamic performance curves were in acceptable agreement. The flow fields and energy losses were investigated in stage planes with the help of pressure 5 hole conical probes. The obtained experimental spanwise distributions of rotor and stator loss coefficients were compared with computed ones at design point.

Then the complete axial flow fan with stage diameter of 600 mm was investigated on test rig built up according to ISO standards. Theoretical relations derived on 1D flow analysis in complete fan showed that fan efficiency decreases with energy losses of inlet chamber and diffuser and with square of flow coefficient. This tendency is weakened by the increase of stage pressure coefficient. These conclusions were confirmed by experimental and flow simulation results. At design point the difference between experimental stage blading and fan efficiency values was 7- 8%.

Comparison of experimental and computed aerodynamic performance of complete axial flow fan showed good agreement at three typical working points. A basic variant of inlet chamber with seven struts was used in order to obtain good stiffness of bearing support. Struts in lower part of inlet chamber were longer and thicker than in the upper part. An analysis of flow mechanism in inlet chamber and diffuser was carried out on the basis of flow simulation data. Energy losses of these fan parts did not significantly change with volume flow rate variation. The flow simulation results obtained in axial flow fan with 3 variants of inlet chamber have shown that the decrease of number, length and thickness of struts increases fan efficiency value.

cyrus.aht@iol.cz

EXPERIMENTAL INVESTIGATION OF DUCTED COUNTER-ROTATING AXIAL FLOW FANS

Hussain NOURI / Arts et Métiers ParisTech - Laboratoire DynFluid (France)
Florent RAVELET / Arts et Métiers ParisTech - Laboratoire DynFluid (France)
Farid BAKIR / Arts et Métiers ParisTech - Laboratoire DynFluid (France)
Christophe SARRAF / Arts et Métiers ParisTech - Laboratoire DynFluid (France)

An experimental study on counter-rotating axial-flow fans was carried out. The fans of diameter D= 375 mm were designed to match the following specification point : Q=3600 m3/h, total pressure rise ΔPt= 420 Pa rotating at 2000 RPM (specific speed Ω=2.46). The two rotors have been designed using the MFT software developed by the DynFluid Laboratory and based on the inverse method. The counter-rotating fans operate in a ducted-flow configuration and the overall performances are measured in a normalized test bench. The rotation rate of each fan is independently controlled. The axial spacing between the fans can vary from 10 to 150 mm.

The results show that the efficiency is strongly increased compared to a conventional rotor or to a rotor-stator stage. The effects of varying the rotation rates ratio on the overall performances are studied and show that the system is highly efficient on a wide range of flow-rates and pressure rises. However, the change of the axial distance between rotors from 10 to 50 mm does not seem to change the overall performances. This system has thus a very flexible use, with a large patch of high efficient operating points in the parameter space. On the other hand, wall pressure fluctuation measurements on the casing wall, 5 mm downstream the front rotor were performed. These measurements reveal tendencies about the overall machine noise and the rotors interaction. Results show that there is a strong interaction between rotors at S=10 mm with a richer power spectrum including interaction frequencies indeed the blade passing frequencies of each rotor. However, this interaction is highly decreased when the inter-rotor space is S=150 mm with still relatively high performances. The cross-correlation and auto-correlation of the wall-pressure fluctuations highlight the presence of coherent structures in between the fans, highly correlated and rotating at the same frequency and direction than the blade passing frequency of the rear fan when S=10mm. At S=150 mm however, the flow is less correlated but a rotating structure is still observed, rotating at the blade passing frequency of the front rotor. From these local measurements one could imagine that noise reduction could be expected when the axial spacing S is increased with relatively high performances.

Local measurements of the velocity field in-between the fans are scheduled soon, in order to compare the predicted and measured flow velocity and angle along the blades at the fans exit.

hussain.nouri@paris.ensam.fr

CFD SIMULATIONS TO PREDICT THE ENERGY EFFICIENCY OF AN AXIAL FAN FOR VARIOUS CASING CONFIGURATIONS

Alain GUEDEL / CETIAT (France)
Mirela ROBITU / CETIAT (France)
Vivian CHAULET / Ziehl-Abegg FMV (France)

The implementation of the Energy Related Products Regulation applied to fans (ErP Lot 11), which is now entered into force, fixes minimum energy efficiency requirements for fans from 1 January 2013. The target energy efficiency is based on the electrical input power of the motor at the best efficiency point of the fan, either static or total overall efficiency according to the installation category.

Fan manufacturers may have difficulty to comply with this European regulation, which forces them to improve some of their fans. To help manufacturers in this task, the objective of this paper is to assess the ability of a CFD numerical method (stationary RANS equations with standard k-ε turbulence model) to predict the fan impeller efficiency with reasonable accuracy.

The study has been carried out on a low-pressure axial flow fan of 630-mm diameter, 12 blades, hub/tip ratio of 0.33, running at 1450 rpm. Various casing configurations such as long or short casing, with or without inlet bellmouth, have been studied with the same impeller design. The influence of the installation category A or D has also been examined in this work. The use of flow simulations is indeed beneficial to predict the performance of variants of an initial design as it is generally quicker and less expensive than making tests on various test set-ups, especially when the number of variants is important.

The CFD calculations are compared to the measured performance and efficiency curves for all the tested configurations. The agreement between the numerical and experimental results is satisfactory for the purpose of the present work. Detailed information on the simulations and experiment will be provided in the paper.

alain.guedel@cetiat.fr

CFD MODELS FOR THE ANALYSIS OF ROTOR-ONLY INDUSTRIAL AXIAL-FLOW FANS

Massimo MASI / University of Padova (Italy)
Andrea LAZZARETTO / University of Padova (Italy)

The actual performance of industrial fans has not yet reached the level claimed as possible by a considerable amount of up-to-date literature in which organized campaigns of CFD calculations are performed using optimisation algorithms. On one side this is due to a well-established know-how often considered as "satisfactory" by the manufacturers. On the other side there is still a lack of confidence in the technical and economic advantages deriving from time consuming optimisation techniques.

In this context, local and total performance of an actual rotor-only axial flow industrial fan have been evaluated by experimental tests and CFD calculations. Different degrees of the overall model complexity and amplitudes of the computational domain are considered in the latter to find the best compromise between accuracy of the results and saving of computational effort and costs. In particular, 4 different computational domains were analysed:

- A single blade passage without tip clearance and idealized meridional geometry;
- A single blade passage and idealized meridional geometry;
- A single blade passage with tip clearance and actual meridional geometry (including bell-mouth entry, electric motor and other ancillary components);
- Entire fan within an idealized fan test-rig assembly.

All computational domains were discretised taking advantage of the innovative polyhedral grid meshing capability of CFD commercial codes. The importance of the proper grid size for scientific and industrial applications is discussed also accounting for the turbulence model selection. When the analysis is carried out at the maximum level of detail, more than 1,500,000 cells are needed to get reliable results for each fluid volume containing the impeller blade channel. When, instead, industrial applications are considered meshes with 600,000 cells are acceptable if two-equations one-layer turbulence models are used.

Results show that single blade channel stationary calculations with an idealized meridional passage show the better predictions of the total fan performance and efficiency in the stable operation range, also because of a "compensation" between the excessive dissipation that is predicted and the geometry idealization. Instead, reliable analysis of the rotor flow field can be performed only by modeling in detail the actual meridional geometry. In addition, calculations of the entire fan domain on a very rough grid show capability of performance and efficiency prediction that appears very attractive for fan industry.

massimo.masi@unipd.it

ACOUSTIC CHARACTERIZATION OF A CENTRIFUGAL FAN IN AN AUTOMOTIVE VENTILATION UNIT SOUND PREDICTING MODEL

Ali AL MEZZAWI / CETIM & UTC (France)
Yvon GOTH / CETIM (France)
Xavier CARNIEL / CETIM (France)
Olivier CHERIAUX / Valeo (France)

The acoustical model used to predict the sound generated by the centrifugal fan of automotive HVAC systems is described in this paper. It is one of the ventilation unit components, considered as main noise sources, that were modelled in an approach for automotive HVAC sound prediction based on virtual prototyping and noise synthesis. The other noise sources studied are the heat exchanger and the butterfly flap.

A correlation found in the ASHRAE Handbook was used for the centrifugal fan model. It relates the sound power level at the nominal working point of a fan to its aerodynamic flow rate and its pressure rise. Our correlation was developed in order to calculate the sound power at the actual working point. Results obtained by the ventilation unit sound prediction model has allowed to define the ranges of frequencies and flows, related to the flap opening, where the fan is the dominant source.

Sound power measurements were performed on a prototype of an automotive ventilation unit , containing only the investigated components. These measurements were done under the same hypothesis applied to the prediction model. They show that when the fan is the dominant source, the predicted and the measured values are very similar.

This work was performed within the REVA/CESAM project. Involved Partners are Valeo Thermique Habitacle, Université de Technologie de Compiègne, ESI Group and CETIM.

ali.almezzawi@cetim.fr

CETIM_VENTIL: A SOFTWARE TO PREDICT THE NOISE OF A COOLING FAN UNDER A HOOD

Yvon GOTH / CETIM (France)

One of the main acoustic sources under the engine hood of the agricultural and building machines is the fan associated with the cooling system. In this case, an acoustic optimization of the system cannot be dissociated from a thermal optimisation.

In a technological context where the power of the machines tends to increase in a space which cannot evolve any more, and where the cooling system is increasingly complex, the global optimization of the system on intuitive bases becomes increasingly difficult. The numerical modelling tools then become essential.

Upstream of the generic computational aeroacoustics software, very powerful but also very expensive in use, remains a place for simpler modelling tools, with limitations on the configurations which they can handle, but easier to use. One presents such a numerical tool, elaborate by associating software elements already existing in Cetim, and by integrating some of the experimental results obtained on former studies. This software has been defined to be usable by industrial design departments, and to allow taking into account the environment effect and the thermal efficiency on the acoustic performances of a cooling fan confined under a hood.

yvon.goth@cetim.fr

AEROACOUSTICAL INVESTIGATIONS ON AXIAL FANS FOR AUTOMOTIVE COOLING SYSTEMS

Mohamed ZAYANI / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
Şaban ÇAĞLAR / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)
Martin GABI / Institute of Fluid Machinery, Karlsruhe Institute of Technology (Germany)

The noise emission of axial fans for automotive cooling systems has considerable contribution to the sound emission of the vehicle. The target of many investigations is to find measures to reduce the noise of the fan. For this purpose, an understanding of the different noise generating mechanisms is essential. To analyze the influence of important characteristic parameters, like gap shape, the number of blades, the blade spacing and skewed blades, time equivalent measurements of the noise propagation were carried out. The main focus is the change of the geometry while keeping the blade loading constant. The scope of this project is to investigate the influence of skewed blades to the aerodynamic and acoustic behaviour of axial fans. Therefore axial fans with different skewed geometries were built and measured. The variations include unskewed, backward and forward skewed blades. The strength of the skew is also a variation made in this approach and include different form for forward skewed axial fans. The aerodynamic comparison between the different skewed fans shows a very satisfactory result, as the prescribed goal, same aerodynamic condition, has been achieved. All the measured characteristic graphs of the different fans show very small difference, especially in the range of the best point and for a given system curve. The comparison of the efficiency shows an increase by the backward skewed fan in comparison to the unskewed fan and the small displacement of the best point to higher volume flow coefficient. In the case of the forward skewed fans the efficiency decreases with the increase of strength of skew, while the point of maximum efficiency moves to smaller volume flow. The comparison of the acoustic graphs shows the best result with the strong forward skewed axial fan. To explain the lower efficiency in the case of the forward skewed axial fan and the best result in the noise propagation, numerical investigations were carried out. The simulated characteristic aerodynamic curves of the different variants show a good agreement with the measured graphs. Especially the trends between the unskewed and different skewed axial fans are very well reproduced.

zayani@kit.edu

NOISE CONTROL FOR COOLING FANS ON HEAVY VEHICLES

Sabry ALLAM / KTH - Competence Centre for Gas Exchange (CCGEx) (Sweden)
Mats ÅBOM / KTH - The Marcus Wallenberg Laboratory (MWL) (Sweden)
Lars WESTER / Sontech AB (Sweden)

An important subsystem in surface transportation vehicles; rail bound, automotive and heavy duty is the engine cooling module. For this reason the European Commission is funding the Efficient Cooling Systems for Quieter Surface Transport (ECOQUEST) project and the overall technical objectives are innovative contributions towards reduced noise emission and energy efficient cooling modules. There three main objectives in the project concern; implement an integrated simulation platform to gather modelling tools for noise, near-field acoustic scattering and propagation towards listeners in the far field, develop thermally and acoustically optimal cooling modules, and development of innovative fan designs and new passive noise control.

Throughout this project two different main objects for fan passive noise control have been examined both experimentally and theoretically; the heat exchanger and inlet parallel baffle silencers.

For the first object seven heat exchangers were experimentally assessed, using a modified version of ISO 15186-1:2000, to test the acoustic transmission for a diffuse field. In addition a sample from each heat exchanger type was also cut out and tested by measuring the acoustic two-port in a duct, i.e., the transmission and reflection at normal incidence were determined. Theoretically, the basic configuration is assumed to be a matrix of parallel and rectangular narrow channels. The developed model is based on a so called equivalent fluid for an anisotropic medium. It is mainly dependent on the heat exchanger geometry combined with the Kirchhoff model for thermo-viscous wave propagation in narrow tubes. The proposed model is validated using the experimental results.

In order to reduce the transmission through heat-exchangers they can be fitted with parallel baffle silencers. In ECOQUEST a new type of such silencers using Micro Perforated Plates (MPP:s) have been designed and tested. Results from this work are presented showing that such MPP baffle silencers can provide up 10-20 dB added damping in the frequency range of interest.

allam@kth.se

AN INITIAL ASSESSMENT OF THE CHANGES THAT WILL APPEAR IN A FORTHCOMING (2012) REVISION OF EN 12101-3

Geoff SHEARD / Fläkt Woods (United Kingdom)
Neil JONES / NMJ Consultancy (United Kingdom)

The standard EN12101-3:2002 defines the design criteria and testing methodologies necessary to verify that powered ventilators (fans) are capable of extracting smoke and hot gasses in the event of fire. It was developed, in response to a mandate from the European Commission, and enables emergency smoke control fans be tested and certified in accordance with the Construction Products Directive's conformance requirements for Fire Safety equipment. Not long after the official publication of EN12101-3, a working group was convened in May 2004 to review the standard. It had been noted that French and German translations were not always faithful to the original English text. Additionally, as the standard was being applied, technical problems, omissions and ambiguities became apparent. There was insufficient information for manufacturers and assessors regarding factory production control requirements and insufficient guidance given to test laboratories on test setups for various fan types and applications. The standard provided no practical information on the assessment of "minor changes" to approved products, making it difficult for test laboratories to decide if changes were "minor" with, no re-test required, or "major" with additional testing needed, to approve the change.

The working group was not convened to tighten certification requirements. The revision is intended to provide clarification and additional information that will ensure the uniform application of the standard by test laboratories throughout Europe and fully satisfy the requirements of the European Commission's mandate. The revision will significantly expand the guidance on the correct approach to factory production control for assessors, fan manufacturers and motor manufactures. Many technical issues and assessment procedures are addressed, including new items such as the setting up and testing of fans with Variable Speed Drives and Soft Starters.

A provisional draft of the revised standard was completed in May 2010 and issued for public comment. The working group intends to send the completed final draft to CEN before he end of May 2012. Once approved, the revised standard must be translated from English into French and German and its publication is anticipated to take place before the end of 2012.

This paper presents an initial assessment of the changes that will appear in the forthcoming (2012) revision of the standard EN 12101-3. The practical implications of the changes are considered for notified laboratories, fan and motor manufacturers. In so doing the paper clarifies the impact of the forthcoming revision.

geoff.sheard@flaktwoods.com

INVESTIGATIONS ON THE SWIRL FLOW CAUSED BY AXIAL FANS: A CONTRIBUTION TO THE REVISION OF ISO 5801

Philipp MATTERN / Karlsruhe Institute of Technology (Germany)
Sten SIEBER / Karlsruhe Institute of Technology (Germany)
Đorđe ČANTRAK / University of Belgrade - Faculty of Mechanical Engineering (Serbia)
Friedrich FRÖHLIG / Karlsruhe Institute of Technology (Germany)
Şaban ÇAĞLAR / Karlsruhe Institute of Technology (Germany)
Martin GABI / Karlsruhe Institute of Technology (Germany)

Actually the international standard ISO 5801 "Industrial fans — Performance testing using standardized airways" is in revision. In the setups with an "outlet common segment", as defined in ISO 5801, the fan pressure of an axial fan without outlet guide vanes can differ considerably to higher or lower values from the value under duty conditions. This difference depends on the details of the swirl flow at the exit of the impeller with its specific blade design (airfoil, blade shape, vortex design, ...).

The estimation of characteristics at standardized setups with an "outlet common segment" can create a benefit of pressure rise for the fan, but cause also an increase of losses at the flow straighteners, which cannot be corrected with the defined "loss allowances" in ISO 5801. Setups with "outlet duct simulation" or "short duct at the outlet" perform correct measurements of the pressure difference.

To get an impression of the flow and to visualize the vortex which causes this discrepancy High-Speed-Stereo-PIV and pressure measurements were carried out in the downstream tube of an axial fan without guide vains. The HSS-PIV is performed in a cross section as well as in a vertical meridian plane perpendicular to that cross section to visualize the behavior of the vortex further downstream. The intention of the paper is to document and to help to understand the complex flow pattern behind an axial fan. Therefore it delivers detailed information of the velocity component distribution, furthermore comparing them at three different operating points.

The results are presented in movies and time averaged contour plots.

Keywords: axial fan, turbulent swirl flow, High-Speed-Stereo-PIV.

mattern@kit.edu

FLOW CHARACTERISTICS OF BACKWARD CURVED CENTRIFUGAL FAN WITH RECTANGULAR CASING

Hidechito HAYASHI / Nagasaki University (Japan)
Kyohei NAKAMURA / Nagasaki University (Japan)
Souichi SASAKI / Nagasaki University (Japan)
Seiji SHIRAHAMA / Panasonic Ecology Systems Co. Ltd (Japan)
Atsushi NAGATA / Panasonic Ecology Systems Co.Ltd (Japan)

The flow characteristics of the backward curved centrifugal fan with a rectangular casing are investigated with experiments and simulations. The performances of the fans are examined with the experiments. The pressure, the efficiency and the noise characteristics were measured. The simulations are made by the commercial code. The numbers of elements are about 3,000,000 of tetrahedrons. The turbulent model is SST. The steady and unsteady flows were simulated.

At first, the influence of the length of the rectangular casing is examined. The section geometry of one casing is squared and another one is rectangular that is extended with the straight duct. The position of the impeller is centered of the square casing at both ones. It is cleared that the performance of the fan is a little decreased with the casing compared with the impeller only. The total pressure and the efficiency of the fan become low with casing. But they are not influenced with length of the casing. The pressure on the casing varies with the section geometry that the pressure decreased at the narrow passage and increased at the wide passage in the casing. Then the pressure distribution has third harmonic mode for circumferential direction. The pressure out of the impeller varies mainly with the distortion of the net flow that the pressure decreases only on the outlet of the fan, but increases on the opposite of the outlet. Then the distribution has first harmonic mode for circumferential direction. When the net flow out of the impeller increases, the pressure of the fan decreases. That is, the net flow distortion is closely relating to the pressure out of the impeller. The discrete frequency noise of the fan is mainly generated by the interaction of the distorted net flow and the blades.

Second the influence of the position of the impeller in the casing is examined. The position of the impeller was varied to three types for the square casing. The location of the narrow passage in the casing is varied with them. The first one is moved the impeller toward to the inner part of a casing. The second one is moved it perpendicular to the first one. The third one is moved to both directions. The performances of the fan are varied only at the second one. The pressure and the efficiency of second one are increased compared to the original (centered) position. At all cases, the efficiency in the impellers is almost same. But it is varied in the casing at second one. This is caused by the improvement of the flow in the casing.

hidechto@nagasaki-u.ac.jp

TOWARD THE CFD SIMULATION OF SIROCCO FANS: FROM SELECTING A TURBURLENCE MODEL TO THE ROLE OF CELLE SHAPES

Manoochehr DARVISH / University of Applied Sciences - HTW Berlin (Germany)
Stefan FRANK / University of Applied Sciences - HTW Berlin (Germany)

Unstructured grid generators are becoming more popular among engineers, due to the significant amount of time that can be saved by using them. They provide a great flexibility in the treatment of any kind of geometry with minimum user interaction. Selecting an appropriate cell shape from the list of available cell shapes supported by the CFD tool, can have a major influence on the accuracy of the solution and reliability of the results. This should be done considering the physics of the flow, available computational recourses and the complexity of the geometry. Applying a proper mesh configuration to CFD simulations leads to better numerical stability and faster convergence of the solution.

In addition to mesh configurations, selecting a proper turbulence model is another factor which improves the accuracy of a simulation. Turbulence models should be selected according to the computational effort, required accuracy, physics of the flow, etc.

In this study performance curves of Radial fans with forward curved blades, which are also called Sirocco fans are obtained by performing CFD simulations. Sirocco fans are capable of delivering considerably more air volume and also produce higher static pressure than many other centrifugal fan types of the same size and speed(but at the expense of lower efficiencies). Sirocco fans are often used in small furnaces, electronic equipment, ventilation, and air conditioning (HVAC) applications. The lower efficiencies of sirocco fans are due to the inevitable formation of recirculation zones which form between the blade channels of the fan.

In this study, simulation results are presented in two parts, the first includes results of the simulations performed using different turbulence models. These simulations are carried out by using three of the most popular turbulence models i.e. two-equation Realizable k-ε model, two-equation SST k-ω and the one-equation model of Spalart-Allmaras.

Second part includes results of the simulations performed by applying different mesh configurations. Simulations are performed by using four mesh configurations which consist of three unstructured grids that are generated using the same mesh properties by means of different cell shapes i.e. polyhedral cells, trimmed cells and the combination of polyhedral cells (for Rotor) and trimmed cells (for Stator), and a structured grid which is generated by means of ANSYS ICEM mesher.

darvish@htw-berlin.de

EFFECTS OF HOUSING GEOMETRY ON THE PERFORMANCE AND NOISE OF A TWO-OUTLET CENTRIFUGAL FAN - A NUMERICAL STUDY

Ian Y. W. WONG / The Hong Kong Polytechnic University (China)
Randolph C. K. LEUNG / The Hong Kong Polytechnic University (China)
Anthony K. Y. LAW / Raymond Industrial Limited (China)

Noise is a common nuisance of domestic products that contain fans. Centrifugal fans are commonly used in these applications, especially in household appliances and lifestyle products. Centrifugal blowers are able to obtain high pressure, large flow rate, relatively low noise and small dimensions compared to the other types of air-moving devices. However, depending on the different functions imposed by the customer needs, the fan designs of these products is very diverse. Usually the flow within a centrifugal fan is highly three dimensional and unsteady. Therefore, analysis using computational fluid dynamics (CFD) software is frequently adopted to uncover the key noise-generating flow dynamics within a centrifugal fan. In the present study, a centrifugal fan design with two flow outlets is investigated. This two-outlet centrifugal fan design is aimed at providing high mass flow rate but very low noise level, which is suitable for air-purifier applications in quiet premises such as bedroom.

Two dimensional unsteady flow simulations with CFD code, Fluent 6.3, are carried out to analyze the fan flow dynamics and assess its subsequent noise radiation. The calculations were done by solving the unsteady Reynolds averaged Navier Stokes (URANS) equations in which effects of turbulence were accounted for using k-ε model. Although a two dimensional simulations are adopted in the study, all relevant key flow dynamics and key fan performance parameters calculated are consistent with the three dimensional situations. Therefore, the results of the calculations suffices to provide an insight in the dominant noise source mechanisms of the two-outlet centrifugal fan and an indication of the variations of noise level with key fan geometrical parameters, such as the ratio between cutoff distance and the radius of curvature of the fan housing.

Four fan design variations are simulated and compared with existing design. The results of simulations show that the unsteady flow-induced forces on the fan blades are the main noise sources. The blade force coefficients are then used to approximate the dipole source terms in Ffowcs Williams and Hawkings (FW–H) equation for assessing the noise effects. It is found that one design is able to deliver a mass flow 34% more, yet with sound pressure level (SPL) 10 dB lower, than the existing design. This founding is verified by testing at the manufacturer laboratory.

catershoot@hotmail.com

FREE AND SCATTERED ACOUSTIC FIELD PREDICTIONS OF THE BROADBAND NOISE GENERATED BY A LOW-SPEED AXIAL FAN

Korcan KUCUKCOSKUN / Ecole Centrale de Lyon (France)
Julien CHRISTOPHE / von Karman Institute for Fluid Dynamics (Belgium)
Christophe SCHRAM / von Karman Institute for Fluid Dynamics (Belgium)
Michel TOURNOUR / LMS International (Belgium)

This paper deals with the free and scattered acoustic field of the broadband noise generated by a low Mach number axial fan analytically and experimentally. The numerical model proposed is a semi-analytical solution to predict the acoustic field of an industrial fan operating in a turbulent stream. The model employed is based on Amiet's theory on the turbulence-interaction noise for a stationary airfoil. The theory for the stationary airfoil is then applied to the rotating blades of the fan.

However, the model includes far and free-field assumptions, and is derived for spanwise uniform incoming flow properties impinging to the leading edge of the airfoil. For most of the industrial applications and in the present configuration these assumptions then become invalid.

The theory has been therefore extended in order to predict the acoustic response of the airfoil in its geometrical near-field. The extended analytical model is compared with the numerical results and a good improvement is satisfied. The model is also applied to a stationary optimized airfoil located downstream of the turbulent flow in the anechoic chamber. Improved results are obtained in the comparison with the measurements.

In order to take the spanwise varying incoming flow conditions into account a segmentation has been applied.

The improved model has been combined with a Boundary Element Method (BEM) accounting for the acoustic scattering from an obstacle. Due to the statistical characterization of the turbulence, an innovative approach called Acoustic Transfer Vectors (ATV) in the BEM framework is employed for the broadband noise scattering.

The numerical validation of the broadband noise scattering has been performed through a comparison of the results obtained by the extended semi-analytical model combined with the ATV approach with a flat scattering screen, and the analytical model which contains the acoustic scattering from an infinite flat plate.

The model combined with the ATV approach is also applied to an industrial low-speed axial fan operating next to a flat scattering screen. An agreement is observed in the comparison with the measurements in both free and scattered acoustic field.

korcan.kucukcoskun@ec-lyon.fr

BROADBAND NOISE MODELLING AND PREDICTION FOR AXIAL FANS

Michele DE GENNARO / AIT Austrian Institute of Technology G.m.b.H. (Austria)
Helmut KUEHNELT / AIT Austrian Institute of Technology G.m.b.H. (Austria)

Low noise design of HVAC components receives today increasing attention in the frame of automotive, railway and aerospace industry in order to improve the comfort experienced by passengers. Despite of the great progresses done by CFD the assessment of a reliable procedure for numerical simulation of fan noise is still a challenge and sometimes even the adoption of very CPU-demanding approaches (e.g. LES/DES) is not sufficient to provide good results. This is mainly due to the high complexity of the physical phenomena involved which imposes simulation requirements often resulting in a prohibitive computational cost for 3D complex configurations.

The target of this paper is providing a fast, reliable and computationally inexpensive technique for fan noise prediction based on the integration of aerodynamic models for turbomachinery with the semi-empirical aeroacoustic model for aerofoil geometries of Brooks, Pope and Marcolini (BPM) published by NASA in 1989. According to the BPM model the noise generation mechanisms occurring on an aerofoil surface can be classified in three main categories: Turbulent Boundary Layer - Trailing Edge noise (TBL-TE), Laminar Boundary Layer - Vortex Shedding (LBL-VS) noise and Separation Stall (S-S) noise. The BPM model allows estimating the sound pressure level density spectrum in one-third octave band for a generic aerofoil as function of boundary layer integral quantities via an algebraic process based on the correlation of experimental data.

This approach has been implemented in an in-house tool for axial fan and tested on experimental data from literature. The test-case chosen is a 6 bladed fan with a diameter of 299 mm tested at an operative rotational speed of 3000 rpm providing a nominal volumetric flow rate of 0.59 m3/sec. The aerodynamic simulation of this fan has been performed with two approaches:

- Blade Element Theory (BET) coupled with XFOIL
- CFD steady RANS (ANSYS-Fluent) with Multiple Reference Frame (MRF)

The noise spectra computed with the BPM model show a good agreement between experimental and computational data proving that the BPM model is able to capture sound pressure levels and broadband components over a wide range of frequencies (from 0.1 to 5 kHz), both coupled with BET/XFOIL as well as with CFD. Additionally attention is also focused on directivity effect implementation and interfacing issues of the BPM with the aerodynamic models mentioned above.

michele.degennaro.fl@ait.ac.at

VORTEX GENERATION BEHIND THE CYLINDER CASCADE OF FAN GRILL

Matej MILAVEC / Hidria Institut Klima d.o.o. (Slovenia)
Vlado SCHWEIGER / Hidria Institut Klima d.o.o. (Slovenia)
Marko HOČEVAR / University of Ljubljana, Faculty for Mechanical Engineering (Slovenia)
Matjaž EBERLINC / University of Ljubljana, Faculty for Mechanical Engineering (Slovenia)
Branko ŠIROK / niversity of Ljubljana, Faculty for Mechanical Engineering (Slovenia)

The fan guard grill is one of the most important components that contribute to the increased sound power level and dissipation of fluid kinetic energy. Sound power measurements of axial fan operation with and without guard grill have shown differences in frequency spectra between 800 Hz and 2000 Hz, presumably caused by Karman vortex generation. To analyse Karman vortex generation air flow around cascade of cylinder was investigated. Experimental methods included visualization with oil vapour as a polutant and measurements of sound pressure levels in the far field.

Experimental work was performed on the standardized measuring station for aerodynamic and acoustic measurements. The cylinder cascade was mounted on the opening of reverberation room. During the measurement, air adjustment and quantity of oil vapour was altered, accordingly. Experimental installation enabled us the possibility to have similar working and ambient conditions for sound pressure level measurements and experimental visualization with oil vapour.

In addition, numerical simulation of air flow around cylinders was performed. Because of increased turbulence and dissipation of fluid kinetic energy, LES turbulent model was used. Boundary conditions of numerical model were adopted from the measurement results. The numerical model was also used for sound pressure level calculation.

Analysis focused on Karman frequency peaks behind cascade of cylinders and -5/3 dissipation law of fully developed isotropic turbulence. Results confirm that Karman vortex generation is responsible for dissipation of fluid kinetic energy, while results of theoretical, experimental and LES methods agree fairly well. The most advantage of experimental visualisation, sound pressure level measurements and LES numerical model calculation is that they do not disturb observed fluid motion. This fact and good correlation between mentioned technics and theoretical law is base for further work on fluid kinetic energy dissipation into rotating region of axial fans.

matej.milavec@hidria.com

FIN FAN VIBRATION REMEDIATION

Richard M. BALDWIN / Southwest Research Institute (United States of America)
P. Joe PANTERMUEHL / Southwest Research Institute, retired (United States of America)
Venkat K. PILLAI / Kuwait National Petroleum Company (Kuwait)

A major refinery has many fin fan coolers, in this case approximately 760 fans of similar structural design with significant variations. Approximately 30 of them experienced high vibrations and several had broken loose from their overhead supports. Plant personnel had monitored vibrations and attempted to stiffen some of the structures with bracing to reduce vibrations, but with only limited success. Amplitudes were not reduced to satisfactory levels.

A testing and analysis program was initiated to identify the vibration mode shapes, frequencies, and amplitudes. The differences among the various fan designs were evaluated, and 23 of the unsatisfactory fans were tested. The primary tests were operational deflection shapes (ODS) and impact modal tests on non-operating units. The field testing also identified that some of the fans were limited in their ability to push air through the radiators, apparently due to plugging or over design. The air was found to be recirculating back around the outer edge of the fan blades and exiting in a counterflow direction, generating turbulence to excite vibrations.

In addition, a finite element model was constructed of the primary fan design to identify the modes at the frequencies measured in the field. This model was also instrumental in identifying effective modifications to remove or control the identified modes of vibration.

Three problem modes were identified. Two of these were clear in the field data and were the modes addressed by the initial plant modifications. The finite element analysis, however, found a third frequency that was hidden among the other modes. Treatment of this mode made it possible to effectively and adequately control the vibrations of all problem fans by adding structural bracing between fan structures, both laterally and longitudinally.

rbaldwin@swri.org

FAILURE ANALYSIS OF IMPELLER BLADE CRACKING IN A MECHANICAL VAPOUR RECOMPRESSOR (MVR) FAN

Brian DONOHUE / University of Canterbury (New Zealand)
Milo KRAL / University of Canterbury (New Zealand)

A Mechanical Vapour Recompressor (MVR) fan installed in a dairy processing plant has a problem with cracking impeller blades. The impeller is a centrifugal type with a two sets of equally spaced blades – one set spanning from inlet to discharge and the other a shorter intermediate set. The shorter blades exhibited failure from two different fatigue regimes over a two year operating period. The first failure, occurring after approximately 150 hours of service following commissioning, was due to oscillatory motion of the impeller in the second mode of torsional vibration, excited by harmonics of the variable speed drive controller. The second failure was due to coupled bending and first mode torsion, due in part to radial clearance in a rolling element bearing. The crack growth mechanisms for both failures were determined by electron microscopy, the first failure being due to a high load - high cycle environment and the second due to high load - low cycle environment. This paper describes the steps taken to identify the possible mechanisms for loading, the testing undertaken to determine system variables, and presents recommendations for avoidance of further failures.

brian.donohue@canterbury.ac.nz

25 YEARS AGO, ACTIVE CONTROL FIRST APPEARED AS A GREAT SOLUTION FOR FAN NOISE: WHAT HAS HAPPENED?

Steve WISE / Wise Associates (United States of America)

The author first installed an active fan exhaust silencer at an industrial plant in 1987. There are now 11 units at the site, still operating, thus answering one of the fundamental questions at the time – that they are reliable. The superior low-frequency noise reduction, especially dramatic on tonal spectra, was clearly demonstrated. Compared against the alternative traditional passive silencers, the 1” (250 Pa) pressure loss savings leads to reduced fan power consumption that has recovered the original installation cost many times over.

In addition to reliability concerns, there are other factors that suppressed the wide application over the ensuing years: 1) the cost was high and the energy benefits not well appreciated; 2) initially, limited application know-how; 3) less stringent guidelines for acceptable noise levels – specifically a disregard for the annoyance of low-frequency noise and especially pure tones; and 4) now-expired fundamental patents in the hands of a few small companies who failed to sustain long enough for market success.

Along with the present low cost of the electronics, evolving market factors have increased the value of the benefits such that it is time for a fresh look at widespread use of this technology, especially in HVAC applications.

This paper highlights some of the 1000 installed units with which the author has direct familiarity, both industrial and HVAC, answering questions about performance predictability along with lessons learned. The systems over the years have provided exceptional low-frequency noise reduction while cumulatively saving approximately 30 million kW-hrs power consumption, with respective reduction of 20-35,000 tons of CO_2. Low-noise, high-efficiency fan installations are an attainable goal for the future.

stevewise@att.net

IMPLEMENTATION OF ACTIVE NOISE CONTROL INTO DIFFERENT FAN APPLICATIONS

Marc SCHNEIDER / ebm-papst Mulfingen GmbH & Co. KG (Germany)
Ralf BALBACH / ebm-papst Mulfingen GmbH & Co. KG (Germany)

Axial and radial fans used for instance in various air conditioning systems, often are annoying due to their noise emissions.

This paper presents the implementation of Active Noise Control (ANC) into different HVAC fan applications with the objective of noise reduction. Therefore a commercial broadband feedforward ANC system (S-Cube Development Kit, Silentium) is used, which comprises a reference microphone, a loudspeaker, an error microphone and a control unit in which an adaptive algorithm is implemented. Due to the application of the virtual microphone theory, the error microphone is only needed during the calibration process of the system, where acoustic transfer functions are measured.

The present paper deals with the application of ANC to different systems containing a diagonal fan of axial type with 172 mm diameter and a radial fan of 566 mm diameter. ANC was used on the suction side as well as on the pressure side. The achieved noise reduction varies considerably for the different fan applications. In the best case the overall sound power level is reduced by about 6 dB(A) with decreasing certain frequency peaks like the blade passing frequency by up to 17 dB. In the worst case nearly no acoustic benefit by means of ANC could be found.

The achievable noise reduction depends on several factors. For the application of ANC the noise spectrum should have maximum levels in the lower frequency region up to 1000 Hz. For an effective noise reduction a combination of passive and active acoustic methods is important. The passive means absorb the high frequency sound of the spectrum, while the active methods reduce the low frequency noise. Good ANC performance results from good coherence between the two microphone signals, which often is difficult to achieve, because on the one hand the reference microphone should measure the characteristic sound, on the other hand it shouldn't be positioned directly in the flow because of flow disturbances which could affect the microphone signal. If an ANC system is implemented into an application it needs a certain installation space for maximum noise reduction. Often this space isn't available - so a compromise between available space and noise reduction has to be found. Eventually ANC reduces just air borne noise - structure borne noise at other locations of the device can't be reduced.

marc.schneider@de.ebmpapst.com

FAN NOISE ANALYSIS USING MICROPHONE ARRAY

Olivier MINCK / MicrodB (France)
Nicolas BINDER / ISAE (France)
Olivier CHERRIER / ISAE (France)
Lucille LAMOTTE / MicrodB (France)
Valérie BUDINGER / ISAE (France)

Fan noise source localization is currently a significantly investigated topic. Growing demand for comfort and various standardizations imposes to manufacturers to reduce the noise of their equipment or facilities. Unfortunately the fan is often one of the noisiest components. For a better understanding of the noise source generation, MicrodB has developed a localization algorithm for the analysis of rotating sources.

When considering the rotating sources, software makes the calculation grid rotate to be synchronous with the blades. Beamforming is done in temporal domain, for each sample, calculation point location is used to estimate the delay applied on the signal. It results in a temporal signal with irregular sampling frequency. Those signals are then interpolated on regular time scale.

Classical time domain beamforming would consist in a summation of the delayed signals from each microphone, but in MicrodB algorithm presented in this paper, they are swapped in frequency domain, used to build the cross spectral matrix, and the diagonal of this matrix is deleted.

Validations of this algorithm have been done with a tonal source. It has been successively characterized with and without rotation, which allows the localization and level estimation validation. This method is being tested for the analysis of an industrial high speed fan, at the ISAE laboratory.

olivier.minck@microdb.fr

LIST OF EXHIBITORS

Advance Design Technology - Delegate Bag Sponsor and Exhibitor

Dilke House
1 Malet Street
London, UK
WC1E 7JN
www.adtechnology.co.uk

Lorenz Bossi
Tel: +44 (0)20 7299 1170
Email: l.bossi@adtechnology.co.uk

AMCA – Exhibitor

30 West University Drive
Arlington Heights
IL 60004
USA
www.amca.org

Mark Stevens, Deputy Executive Director
Tel: +01 847 704 3290
Email: mstevens@amca.org

Boldrocchi - Exhibitor

Head Office
Viale Trento e Trieste, 93
20046 Biassono
Milan
Italy
www.boldrocchi.it

Boldrocchi France
1114 RN 85
38 300 Nivolas Vermelle
France

Paolo Saccenti
Tel: +39 039 2202500
Email: Saccenti@boldrocchi.eu

Nicole Ranchin
Tel: +33 (0)4 7419 0032
Email: nicole.ranchin@boldrocchi.fr

Brüel & Kjær Sound & Vibration Measurement A/S – Exhibitor

Skodsborgvej 307
DK-2850 Nærum
Denmark
www.bksv.com

Tel: +45 7741 2000
Email: info@bksv.com

CD Adapco - Exhibitor

Paris Office
31, rue Delizy
F-93500 Pantin, France
www.cd-adapco.com

Tel: +33 (0)141 83 7560
Email: info@fr.cd-adapco.com

CETIAT – Organiser and Exhibitor

Centre Technique des Industries Aérauliques et Thermiques
Domaine Scientifique de la Doua, 25 avenue des Arts
BP52042
69603 Villeurbanne Cedex
France
www.cetiat.fr

Service commercial
Tel: +33 472 44 49 00
Email: commercial@cetiat.fr

Cetim – Organiser and Exhibitor

CETIM/IBV
52 Avenue Félix
Louat
BP 80067
60304 Senlis
France
www.cetim.fr

Patrick Vincent
Tel: +33 4 44 67 30 89
Cell: +33 6 86 45 97 87
Email: patrick.vincent@cetim.fr

CFturbo Software & Engineering GmbH – Exhibitor

Headquarters
Unterer Kreuzweg 1
D-01097 Dresden
Germany
www.cfturbo.com

Tel: +49 351 40 79 04-79
Tel: +49 351 40 79 04-80
Email: sales@cfturbo.de

Munich Office
Friedrichstrasse 20
D-80801 Munich
Germany
www.cfturbo.com

Tel: +49 89 189 41 45-0
Tel: +49 89 189 41 45-20
Email: sales@cfturbo.de

Exa Corporation – Gold Sponsor and Exhibitor

Exa Corporation
55 Network Drive
Burlington, MA USA

Tel: +1 781-564-0200
Email: info@exa.com

Exa France - EuroXA SARL
Les Collines de l'Arche, 76 Route de la Demi-Lune
92057 Paris la Défense Cedex, FRANCE

Tel: +33 1-44-45-66-00
Email: france-info@exa.com

Fläktwoods – CD Proceedings and Abstract Book Sponsor

Axial Way
Colchester
Essex
CO4 5ZD, UK
www.flaktwoods.com

Tel: +44 (0)1206 222 555
Email: marketing.uk@flatkwoods.com

Free Field Technologies - Exhibitor

Free Field Technologies
Rue Emile Francqui 1
1435 Mont-Saint-Guibert
Belgium

Julien Manera
Email: Julien.manera@fft.be
Tel : + 00 32 10451226

Howden – Platinum Sponsor and Exhibitor

Howden
Old Govan Road
Renfrew
PA4 8XJ
United Kingdom

Tel: +44 141 885 7300
Email: fan2012@howden.com

IMechE – Organiser and Exhibitor

1 Birdcage Walk
Westminster
London, UK
SW1H 9JJ
www.imeche.org

Tel: +44 (0)20 7222 7899
Email: enquiries@imeche.org

LMS International – Silver Sponsor and Exhibitor

LMS France
2 rue René Caudron
78960 Voisins le Bretonneux
France

Elodie Lauret
Marketing Manager
Tel: +33 01 34 52 17 40
Email: marie-elodie.lauret@lmsintl.com

Yohann Mesmin
Technical Manager Simulation
Tel: +33 01 34 52 17 40
Email: Yohann.mesmin@lmsintl.com

NOTES

NOTES

NOTES

NOTES

NOTES

NOTES