ECCOMAS

ECCOMAS Congress 2024,

3 – 7 June, Lisbon, Portugal

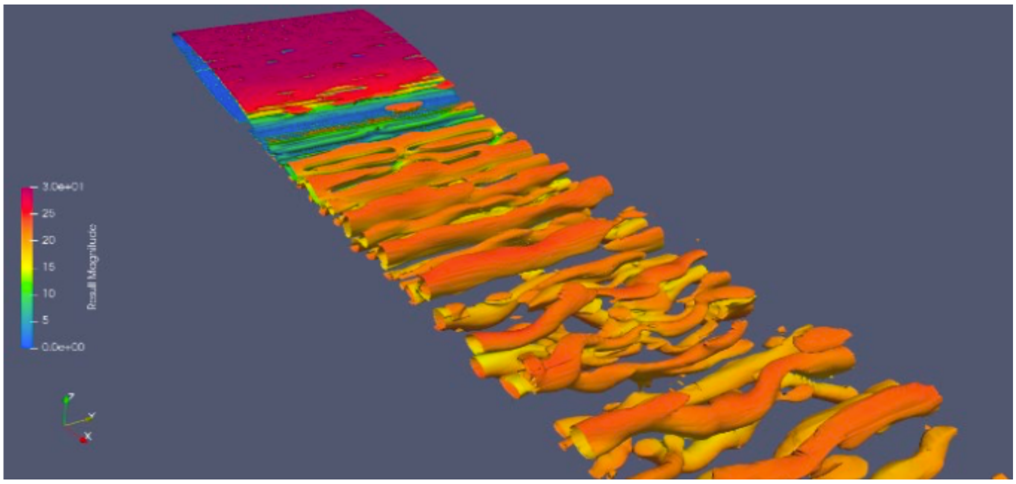
Editors:

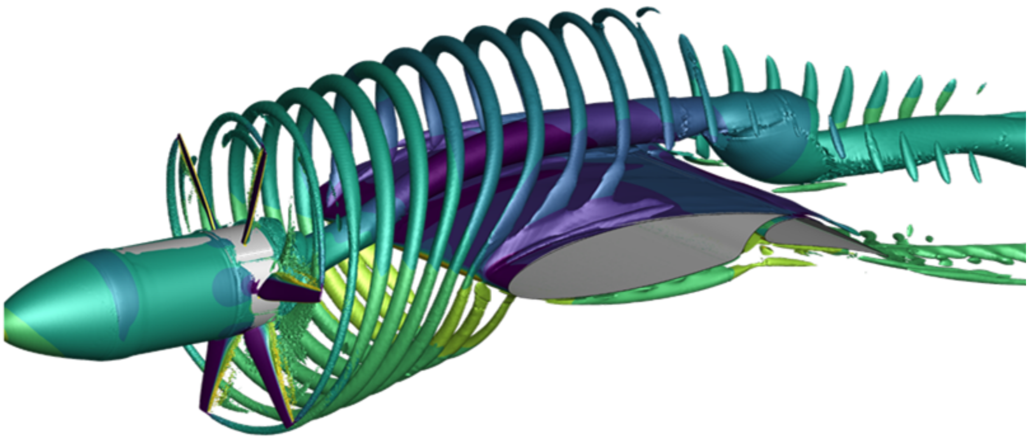
Gabriel Bugeda

Jacques Periaux

Tero Tuovinen

Dietrich Knoerzer





Green and Digital Technologies for Climate, Energy and Transport Challenges

Special Technology Sessions – Book of Abstract

CIMNE INTERNATIONAL CENTRE FOR NUMERICAL METHODS IN ENGINEERING

Industry Interest Group ECCOMAS

jamk University of Applied Sciences

Editors:

***Gabriel Bugeda**, CIMNE / UPC, Barcelona*

***Jacques Periaux,** CIMNE, Barcelona*

***Tero Tuovinen**, JAMK Univ. of Applied Sciences, Jyväskylä*

***Dietrich Knoerzer**, Aeronautics Consultant, Brussels/ Aachen*

Cover Figure 1: STS247-2 - *Three-dimensional Numerical Simulations around an A320 Morphing Wing at High Reynolds Number* by Clément Rouaix, Abderahmane Marouf, Yannick Hoarau, Mateus Carvalho, Horia Hangan, Marianna Braza

Cover Figure 2: STS240-3 - *Advanced CFD Applications for Complex Aircraft Configurations* by Jonas Oldeweme, Till K. Lindner, Christoph Bode, Peter Scholz and Jens Friedrichs

Green and Digital Technologies for Climate, Energy and Transport Challenges
Special Technology Sessions – Book of Abstracts.
Eds. G. Bugeda, J. Periaux, T. Tuovinen, and D. Knoerzer. *In conference*: ECCOMAS Congress 2024, 3.-7.6.2024, Lisbon, Portugal. ISBN 978-84-127483-6-9, (2024).

ECCOMAS Congress 2024

Lisbon, 3 – 7 June 2024

Green and Digital Technologies for Climate, Energy and Transport Challenges

Special Technology Sessions – Book of Abstracts

Overview of the Congress Programme

The table below shows the Special Technology Sessions STS allocation in the Technical Programme.

ECCOMAS Congress 2024, Lisbon, 3 – 7 June 2024 Schedule of the Special Technology Sessions						*Status: 16/05/2024*
Time	**Sunday 2nd June**	**Monday 3rd June**	**Tuesday 4th June**	**Wednesday 5th June**	**Thursday 6th June**	**Friday 7th June**
7:30 - 8:30		Registration				
8:30 - 10:00		Opening Ceremony ECCOMAS Awards Plenary Lecture 1	Plenary Lectures 2 + 3	Plenary Lectures 4 + 5	Plenary Lectures 6 + 7	COM Plenary Lecture 8
10:00 - 10:30		*Coffee Break*				
10:30 - 12:30		Parallel Sessions	**STS 233** (R.0.07) **STS 272** (R.0.08)	**STS 239** (R.0.07) **STS 272** (R.0.08)	**STS 244** (R.0.07) **STS 234** (R.0.08)	**STS 248** (R.0.07) **STS 245A** (R.0.08)
12:30 - 13:45		*Lunch Time*				
13:45 - 14:30		Semi Plenary Lectures 1 – 4*	Semi Plenary Lectures 5 - 8	Semi Plenary Lectures 9 - 12	Semi Plenary Lectures 13 - 16	Semi Plenary Lectures 17 - 20
14:30 - 16:30		****D. Reckzeh, Airbus*** Parallel Sessions	**STS 237** (R.0.07) **STS 271A**(R.0.08)	**STS 269A** (R.0.07) **STS 240A** (R.0.08)	**STS 243A** (R.0.07) **STS 268A** (R.0.08)	**STS 245B** (R 0.08)
16:30 - 17:00		*Coffee Break*				
17:00 - 19:00	Registration	Parallel Sessions	**STS 267** (R.0.07) **STS 272B**(R.0.08)	**STS 269B** (R.0.07) **STS 240B** (R.0.08)	**STS 243B** (R.0.07) **STS 268B** (R.0.08)	Closure
19:00 - 20:00	*Ice-Breaking Drink*					
20:00 - 22:00					*Congress Banquet*	

Preface

On behalf of the Executive Committee of the ECCOMAS Congress 2024, it is my great pleasure to welcome the organizers and all participants at the Special Technology Sessions (STS) on 'Green and Digital Technologies for Climate, Energy and Transport Challenges', mainly addressing technology topics in sustainable aviation with the key goal of contributing to reducing the emission of relevant areas of aviation, surface transport and renewable energies. Those topics focus on diverse aspects ranging from advanced computational physical and mathematical methods, multi-disciplinary design and optimization, the impact of artificial intelligence, high-fidelity simulation technologies, and wing design to critical technologies and operational aspects. In particular, we are grateful to Prof. Jacques Periaux, Dr. Tero Tuovinen, Prof. Gabriel Bugeda and Dr. Dietrich Knoerzer, members of the Industry Interest Group (IIG) of ECCOMAS, for putting forward all the organization of the STS, which have been an important part of the ECCOMAS Congresses since 1996, playing a significant role in establishing a link between research and industry.

José César de Sá, Chairman of ECCOMAS Congress 2024

Foreword

The initiative for the Special Technology Sessions (STS) at the ECCOMAS Congress 2024 in Lisbon, Portugal and the preparation of this STS Book of Abstracts was undertaken and managed by the ECCOMAS Industry Interest Group (IIG) [1].

The ECCOMAS Congresses and the industry technology-oriented STS have a long tradition, based on the fact, that the industry in Europe had and still has a strong need for advanced computational methods especially in fluid dynamics, structure mechanics and design optimisation. Here the Aeronautics sector was and still is a front runner in the application of ambitious numerical methods. Therefore, numerous Special Technology Sessions address primarily aeronautics research topics already since the ECCOMAS Congress 1996 in Paris. But the STS also provide an emphasis on technologies and research of industrial relevance for surface transport, energy and manufacturing.

For the ECCOMAS Congress 2024, the STS series are collected under the title ‘Green and Digital Technologies for Climate, Energy and Transport Challenges‘, in the view of the increasing demand for technologies and innovation for climate neutral and sustainable operation of future aviation, surface transport and other industries.

On the first day, Daniel Reckzeh, Airbus Technology will deliver a semi-plenary lecture with the title ‘Energy Efficiency - A Key Lever for Sustainable Flight’, as a scene setter for the series of STS.

For the ECCOMAS Congress 2024, numerous senior experts from Europe, China and Japan have organised their STS together with knowledgeable specialists as speakers from industry, research institutions and academia. They all together ensure the high scientific and technical quality of the STS. In total, fifteen (15) STS address relevant technological themes of aeronautics, transport and industry with twenty one (21) sessions with more than 100 papers on four days. This booklet provides the abstracts of the STS and of their contributing papers. It should give guidance to interested participants in finding for them interesting STS contributions. Many papers and even complete STS present the technological achievements of collaborative projects, which were or are supported by the Research & Innovation Framework Programmes of the European Union as Horizon Europe.

The STS Organizing Team would like to thank the Executive Committee of the ECCOMAS Congress 2024, especially its Chairman Professor José César de Sá, for welcoming and integrating the STS in the conference pro-

[1] https://www.eccomas.org/comittees/industry-interest-group-iig/

gramme. Our thanks go also to Daniel Reckzeh for his lecture and to the chairpersons and speakers of the STS, as they ensure the scientific quality by their effort. We acknowledge the support of the CIMNE Congress Bureau acting as the ECCOMAS Secretariat and thank especially Gemma Barberillo Nualart and Alessio Bazzanella for all their help and for integrating the STS in the conference. We hope, the readers will appreciate the overview that this STS Book of Abstracts provides and will use it for finding interesting technological contributions of the industry-oriented STS.

The STS Organizing Team

Gabriel Bugeda, CIMNE / UPC, Spain, bugeda@cimne.upc.edu
Jacques Periaux, CIMNE, Spain, jperiaux@gmail.com
Tero Tuovinen, JAMK Univ. of Applied Sciences, Finland, tero.tuovinen@jamk.fi
Dietrich Knoerzer, Aeronautics Consultant, Brussels, Belgium / Aachen, Germany, dietrich.knoerzer@outlook.com

Barcelona / Jyväskylä / Brussels / Aachen, June 2024

ECCOMAS Industry Interest Group

The ECCOMAS Industry Interest Group (IIG) is a special interest group operating within ECCOMAS. Its main objective is promoting research and innovation (R&I) with industrial relevance. Its activities include strengthening the link to the industry and developing contacts between ECCOMAS and the different R&I related services of the European Commission.

For promoting these interactions, the main activity of the IIG is organising various kinds of events such as thematic conferences and workshops. Since 2016, the IIG organises industry relevant special technology sessions (STS) at the large ECCOMAS Congresses. At these events, technologists from industrial sectors and experts from the scientific community can exchange their experiences and future needs. These conferences and workshops provide also opportunities for learning, future oriented discussions and new initiatives. The IIG also aims to represent a valuable platform for networking crucial for building stronger collaboration.

At the ECCOMAS Congress 2024 in Lisbon, Portugal, the ECCOMAS Industrial Interest Group organises fifteen (15) STS with twenty one (21) sessions, which focused on industrial, environmental and societal challenges in aviation, surface transport and other industries. By this the IIG offers answers to ECCOMAS Congress participants on uprising issues as Artificial Intelligence (AI), innovative digitalized methods, providing numerical solutions in particular for sustainable developments and greening mainly in aviation but also in surface transport.

Contents

Please note that in the case of several paper authors, the presenting author if marked with a star (*) behind his/ her name.

Daniel Reckzeh, Airbus Technology, Semi-Plenary Speaker

Semi-plenary Lecture

Daniel Reckzeh joined the German Aerospace Center DLR and later Airbus after graduating in 1997 in Aerospace Engineering at the Technical University of Braunschweig, Germany. He has taken various leading positions in Airbus Flight Physics, having been involved in the design of the A380, A400M and various Research & Technology Programmes and was playing a driving role in the invention of the A350XWB multifunctional wing concept.
In his current role as Senior Manager in the Airbus Technology Program, Daniel is responsible for Overall Aircraft Design & Integration and Flight Science Technologies. Besides this, he is leading the Research & Technology organisation on the Airbus site in Bremen. Daniel is holding an Airbus Aircraft Architect position and also an Expert role in the technology field of multidisciplinary wing design. He is member of the Senate of the German Aerospace Society (DGLR) and leads the section "manned aircraft".

As a scene setter for the STS, he delivers the Semi-Plenary Lecture on Monday, 3rd June 2024 (SPL04 at13:45 - 14:30 in Auditorium VI):

Energy Efficiency - A Key Lever for Sustainable Flight

Semi-Plenary Lecture SPL04

"Energy Efficiency" - A Key Lever for Sustainable Flight

Daniel Reckzeh

Senior Manager, Airbus Technology, Airbus Commercial, Airbus Allee 1, 28199 Bremen, Germany
daniel.reckzeh@airbus.com

Abstract

Key Words: *Energy efficiency, next generation aircraft, sustainable fuels, air transport system*

Advanced aircraft technologies for low drag, low weight and low specific fuel consumption are more important than ever to master the challenge towards more sustainable flight. Significant efficiency improvements are mandatory to limit the fuel consumption for next generation aircraft, either to drastically reduce the CO2-impact from in transition period still fossil kerosene based propulsion or to mitigate the drawback from the significant affordability challenge of future sustainable fuels (SAF or H2). Besides this efficiency challenge also the capability for more flexible flight trajectories is key, to mitigate non-CO2 impacts on global warming (such as contrails) and to drastically cut noise emissions. Nevertheless, full compliance in terms of certifiability, manageable manufacturing cost and full operational compatibility to the air transport system are crucial requirements to ensure economically viable future products.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 233

Advanced Computational Physics/ Computational Mathematics Methods and Tools for Improving a Climate Neutral Digitalized Transport

Chairs: Jacques Periaux[1] and Gabriel Bugeda[2]

International Centre for Numerical Methods in Engineering (CIMNE)
Edifici C1 Campus Nord UPC, C/ Gran Capità S/N, 08034 Barcelona, Spain,
https://www.cimne.com,[1] jperiaux@gmail.com, [2] bugeda@cimne.upc.edu,

Session Abstract

Key Words: *Aviation CO2 emissions, aviation non-CO2 emissions, cirrus formation*

The results on research and innovation (R&I) in Computational Physics/ Computational Mathematics, which are to be presented in this STS by invited scientists, technologists and meteorologists, will consider the minimisation of aviation CO2 and non-CO2 emissions with following targets:

i) Increase the scientific understanding related to the contribution of aviation Greenhouse gas emissions to climate change;
ii) Increase performances of multi engines distributed propulsion particle emissions characterisation;
iii) Achieve better performances with hydrogen and aviation synthetic fuel reducing further non-CO2 emissions;
iv) Characterize better the contrail formation and effects and provide more insight in the aviation NOx emissions and ozone formation;
v) Develop further real-time decision-support software for airlines and ATM, to predict the location and global warming impact of contrail and contrail cirrus formation.

This STS will provide the audience with innovative methods and tools in different i) -v) sectors recent studies showing the decrease of CO2 – non-CO2 emissions and the cost-effective mitigation measures and computations resolving uncertainty short term or long terms climate impact.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 233-1

Innovative Technologies for Aviation in Support of Europe's Green Deal

Michael Kyriakopoulos*

European Commission, DG Research & Innovation, 1049 Brussels, Belgium
Michael.kyriakopoulos@ec.europa.eu

Key Words: *Horizon Europe, Aviation, Critical Technologies, ACARE, Green Deal*

Horizon Europe is the EU Framework Programme for Research and Innovation, which aims to build a knowledge- and innovation-driven society and competitive economy and contribute to sustainable development also in an important high-tech sector as aviation. The public-private partnership Clean Aviation and the transport related collaborative research address the critical technologies for greening aviation.

In spring 2022 the stakeholders of aviation organised in ACARE – the Advisory Council on Aviation research & Innovation in Europe – published Europe's vision for sustainable aviation under the title 'Fly the Green Deal' [1].

Europe's aviation sector continues its resilient and pioneering spirit as it leads the world's transport system into its new era of great transformation. Surviving the pandemic, it is adapting rapidly to satisfy the rising demand for competitive air mobility services while managing a scarcity of resources and embracing the new challenges of climate change and energy transition. Facilitated by ACARE, the European Commission, its Member States, aviation research organisations, design and manufacturing industries, airlines, airports, and aviation energy and service providers have all joined together to envision a synchronized transformation path that will ensure that Europe can lead the world towards a climate neutral, citizen centric and competitive air mobility system. "Fly the Green Deal" is Europe's vision for sustainable aviation. It describes the actions and actors necessary towards aviation's three main strategic goals. It details three time horizons and defines as well the requirement for a proactive and synchronised implementation framework facilitated by the European Commission and EU Member States that includes both the initiating instruments (policies, regulations and incentives) and a system of measuring and impact monitoring to ensure the goals are achieved.

References

[1] European Commission, Directorate-General for Research and Innovation, Fly the Green Deal – Europe's vision for sustainable aviation, Publications Office of the European Union, 2022, https://data.europa.eu/doi/10.2777/732726

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 233-2

Simulation of Radiative Transfer in Variable 3D Atmosphere for Contrails

Olivier Pironneau[1] and Pierre-Henri Tournier[2]*

[1] Laboratoire Jacques-Louis Lions, CNRS, Sorbonne Université, 4 place Jussieu, 75005 Paris, France, olivier.pironneau@sorbonne-universite.fr

[2] Laboratoire Jacques-Louis Lions, CNRS, Sorbonne Université, 4 place Jussieu, 75005 Paris, France, pierre-henri.tournier@sorbonne-universite.fr

Key Words: *Radiative transfer, climate, open-source software, high performance computing*

The radiative transfer equations have three spatial dimensions, two radiational dimensions, one frequency variable, and possibly time. In this work, without loss of generality, they are reduced to a nonlinear integro-differential system and solved by an iterative scheme which is shown to be monotone and convergent. At each step, two large integrals of convolution type need to be computed. The kernel matrices are assembled in compressed form using Hierarchical matrices, with the help of the H-Matrix high performance parallel library Htool[1]. The whole method is implemented in FreeFEM[2] [3], a free open-source software designed for multi-physics simulations. The resulting method is of complexity n log(n), where n is the number of vertices of the physical domain.

As an example, the method is applied to the temperature and radiation in the valley of Chamonix (France), the challenge being the large variation of density, altitude and light reflection due to snow and clouds. Our method is able to handle 240K physical points, all directions of radiation and 683 frequencies in less than 35 minutes on an Apple M1 laptop [1]. The method has also been extended to handle reflective boundary conditions [2].

References

[1] F. Golse, F. Hecht, O. Pironneau, D. Smets and P.-H. Tournier, Radiative transfer for variable three-dimensional atmospheres. Journal of Computational Physics, 475, 111864, 2023.

[2] O. Pironneau and P.-H. Tournier, Reflective conditions for radiative transfer in integral form with H-matrices. Journal of Computational Physics, 495, 112531, 2023.

[3] F. Hecht, New development in FreeFem++. Journal of numerical mathematics, Vol. 20, no 3-4, pp. 251–266, 2012.

[1] https://github.com/htool-ddm/htool

[2] https://freefem.org/

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 233-3

Multi-fidelity Propeller Design for Low Reynolds Number Operating Regimes

Domenico Quagliarella[1,*], Antonio Pagano[2], Donato De Rosa[3] and Antonio Carozza[4]

CIRA – Italian Centre for Aerospace Research, Via Maiorise 1, 81043 Capua, Italy, https://www.cira.it,
[1]d.quagliarella@cira.it, [2] a.pagano@cira.it, [3] d.derosa@cira.it, [4] a.carozza@cira.it

Key Words: *Low-Reynolds Aerodynamics, Propeller Design, Evolutionary Optimization*

The recent development of drones for very high altitudes or hybrid airships that generate aerodynamic lift in addition to the aerostatic lift typical of airships brought attention to the analysis and design of very low Reynolds number configurations. This work, in particular, focuses on the design issues of a propeller for this type of airship equipped with electric propulsion, which must offer high performance in terms of efficiency in the presence of phenomena such as the transition from laminar to turbulent flow and laminar bubbles. Dealing with these phenomena in the design phase is difficult because their correct prediction requires sophisticated and computationally expensive computational models. The approach followed here is hierarchical and is based on a hierarchy of solvers of increasing fidelity, moving from an integral boundary layer coupled to a potential solver with compressibility correction to approaches based on the coupling of solvers of the Euler or Reynolds Averaged Navier-Stokes equations (RANS) coupled boundary layer with simple transition models, up to RANS solvers with transitional turbulence models such as γ-Re_θ.

The adopted approach starts from the 2D multi-point design of various airfoils for different radial stations of the propeller blade, carried out with an evolutionary approach, and, in a second step, these profiles are used to design the propeller blade.

The developed methodology is demonstrated in a challenging case study in which we will also try to evaluate the margins of uncertainty of the developed design.

References

[1] M. Carreño Ruiz, L. Renzulli, and D. D'Ambrosio. Airfoil optimization for rotors operating in the ultra-low Reynolds number regime. Physics of Fluids, 35(10):103603, 10 2023.

[2] T. Lefebvre, P. Beaumier, S. Canard, A. Pisoni, A. Pagano, A. Sorrentino, B. Van Der Wall, J. Yin, A. D'Alascio, C. Arzoumanian, and S. Voutsinas. Aerodynamic and aero-acoustic optimization of modern tilt-rotor blades within the ADYN project, ONERA-PUBLICATIONS-TP, (158), 2004.

[3] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Völker. A correlation-based transition model using local variables—Part I: Model formulation. J. Turbomach., 128(3):413, 2006.

[4] A. Rausa, A. Guardone, and F. Auteri. Implementation of γ-Re_θ transition model within SU2: model validation and verification. In AIAA SCITECH 2023 Forum, 2023.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 233-4

Gradient-Based Optimization of Turbomachinery to Reach Climate Neutrality

Thanh-Son Tran[1], Luca Zampini[1,*], Nassim Razaaly[2], Lasse Mueller[1], Mohamed Aissa[1], Arnaud Châtel[1], Tom Verstraete[1]

[1] Von Karman Institute, Waterloosesteenweg 72, 1640 Sint-Genesius-Rode, Belgium,
luca.zampini@vki.ac.be, https://www.vki.ac.be

[2] ISAE ENSMA, Téléport 2 – 1 avenue Clément Ader, 86000 Poitiers, France,
nassim.razaaly@ensma.fr, https://www.ensma.fr

Key Words: *Gradient based optimization, adjoint, multidisciplinary optimization*

To reach climate neutrality in aviation, electric propulsion is one of the many different alternatives currently looked into. A recurrent approach in electric propulsion for short haul commercial flight is to apply distributed propulsion along the wing span using small electric driven jets. These electric jet engines use a fan to accelerate the fluid and generate the thrust. The efficiency of the fan is in direct relation to the range of the aircraft for a given installed battery capacity and is therefore crucial for a successful introduction of electrically propelled flight.

This lecture looks into a multidisciplinary optimization framework for turbomachinery that enables the design of highly performant fans. A Reynolds Averaged Navier Stokes solver is integrated with a Computational Structural Mechanics solver to strike a balance between aerodynamic efficiency and structural integrity. Adjoint solvers for both domains have been derived to provide gradient information to a Sequential Quadratic Programming optimizer. The developed optimization strategy is applied to the well-known NASA rotor 37 test case as a prototype for later use in electric propulsion systems.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 233-5

Evolutionary Multidisciplinary Shape Design Optimization of a Disruptive Aircraft Configuration with Distributed Propulsion: Challenges for Climate Neutrality

Zhili Tang[1], Shaojun Luo[1], Haiqin Li[1], Jacques Periaux[2] and Gabriel Bugeda[2,3]

[1] College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China, tangzhili@nuaa.edu.cn

[2] Centre Internacional de Mètodes Numèrics a l'Enginyeria (CIMNE), C/ Gran Capità s/n, Campus Nord UPC, Mòdul C1, Barcelona, Spain, www.cimne.edu, jperiaux@gmail.com, bugeda@cimne.upc.edu

[3] Universitat Politècnica de Catalunya, C/ Gran Capità s/n, Campus Nord UPC, Mòdul C1, Barcelona, Spain, www.upc.edu

Key Words: *Distributed propulsion, Blended Wing Body, Multidisciplinary Optimisation*

Distributed propulsion is used with small or medium-sized engines installed on the rear of the aircraft instead of centralized installed large engines. This disruptive configuration improves the propulsion efficiency by distributing exhaust and filling the velocity deficit due to the wake. It changes the layout of traditional aircraft with centralized installation thrust and alleviates also the high-performance requirements and difficulty to design large engines.

This lecture surveys propulsion efficiency, aerodynamic characteristics, airframe and engine integration, structural weight, flight stability and control, the effect of boundary layer ingestion on fuel efficiency, and active control to reduce distortion of inlet flow, the mechanism for improving thrust and fuel efficiency.

Based on the above analysis, the distribution criteria of the thrust, interference characteristics between jet and wake flow, design optimization methods considering the boundary layer ingestion as well as active flow control tools for improving the distortion of air flow at the inlet of the engine are introduced.

Finally, in the context of climate neutrality and combining the distortion of micro engines, aerodynamic, structural dynamics and flight control, the multidisciplinary shape design optimization of a Blended Wing Body (BWB) aircraft with distributed propulsion is performed with evolutionary computing and results are presented and discussed.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 234

Drag Reduction for Transport Aircraft

Chair: Geza Schrauf

Consultant, Bremen, Germany, contact@schrauf.de

Session Abstract

Key words: *Drag Reduction, Friction Drag, Induced Drag, Laminar Flow, Span Extension*

Both the United Nations climate conference of Paris in December 2015 and Europe's Vision for Aviation 'Flightpath 2050' sets have set a target of 75% reduction of specific fuel consumption by 2050, compared to the standard for civil aviation in 2000. To achieve this ambitious goal, we need to work on three topics: improvement of engine efficiency, reduction of the aircraft weight, and aerodynamic drag reduction.

In this session, we cover some aspects of the last topic, the reduction of aerodynamic drag. The latter consists of two main components: lift-independent (friction or viscous) drag and lift-dependent or induced drag [1]. Both drag components will be addressed in the session.

Regarding friction drag reduction, we present new developments of the application of natural laminar flow. We will consider forward-swept wings, which make it easier to keep the attachment line laminar. Furthermore, we take a look at a backward-swept, laminar wing with a reduced chord which is complemented by a larger span. This approach combines two advantages. The smaller chord-Reynolds number makes it easier to achieve and maintain laminar flow, and the increased span reduces the induced drag.

Then we will present the latest results of Clean Aviation [2, 3] regarding hybrid laminar flow control applied to wing and horizontal tail plane.

In the second part of the session, we address new structural concepts combined with advanced load alleviation, which have a great potential to allow for a lighter wing with increased span, and, thus, reduce lift-dependant drag [4]. A fuel burn reduction of 30 % compared to the state-of-the-art reference Aircraft A321neo is expected.

References:

[1] G. Schrauf: "Status and perspectives of laminar flow." The Aeronautical Journal, December 2005, Vol. 109, No. 1102, pp. 639-644.

[2] Clean Aviation; "HLFC on Tails & Wing." https://clean-aviation.eu/clean-sky-2/demonstrators/key-demonstrators/hlfc-on-tails-wing.

[3] T. Haase, S. Ropte, B. van de Kamp, A. Pohya, M. Kleineberg, A. Schröder, J.-L. Pauly, T. Kilian, J. Wild, U. Herrmann: "Next generation wings for long range aircraft: hybrid laminar flow control technology drivers." Deutscher Luft- und Raumfahrt Kongress, 01.-03. September 2020. virtuell. https://elib.dlr.de/136995/.

[4] Clean Aviation: "UP Wing - Ultra Performance Wing." https://www.youtube.com/watch?v=MjrX6GvPGD8 .

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 234-1

Natural Laminar Flow at Cruise Mach Number 0.78: First Results of ETW Concept Verification Tests

Arne Seitz[1]*, Javier Ruberte Bailo[2] and Thomas Streit[3]

DLR, Institute of Aerodynamics and Flow Technology,
Lilienthalplatz 7, 38108 Braunschweig, Germany, https://www.dlr.de/as/en
[1] Arne.Seitz@dlr.de, [2] Javier.RuberteBailo@dlr.de, [3] Th.Streit@dlr.de

Key Words: *Transport aircraft, Natural laminar flow, Forward sweep, Transonic flow*

Within the frame of DLR internal Project TuLam a short and medium range transport aircraft with a natural laminar flow forward swept wing (NLF-FSW) was designed, [1]. Top level aircraft requirements were taken from the reference aircraft Airbus A320, in particular the design Mach number of MD = 0.78 was kept. The basic idea then was to exploit the beneficial effect of a combination of forward sweep and taper which leads to a wing with low leading-edge sweep (-17° compared to +27° for an A320), while in the recompression zone the sweep is the same as for the backward swept one. Due to the low leading-edge sweep, attachment line transition and cross-flow instability growth can be controlled solely by natural means, i.e. contour shaping. Hence, it was possible to design a wing with the transition line on the upper surface located approximately at the shock position of 65% chord without additional wave drag penalties. In order to verify the NLF-FSW concept, a wind tunnel model was derived from the final wing-body configuration and subsequently tested in the European Transonic Windtunnel ETW, see Fig. 1 (left). Polars were measured at design as well as off design conditions, with the Mach number ranging from low-speed at M = 0.2 to high-speed at M=0.82. All polars were run with transition free on the upper surface as well as transition fixed at approximately 5% chord. Transition detection was performed at selected polar points using the temperature step technique (typical result see Fig. 1). However, Reynolds number for all polars was with 16. Mio only 2/3 of the free flight value in order to prevent turbulent wedges induced by leading edge contamination from small particles in the tunnel as far as possible. Therefore, measured drag polars have to be corrected for free flight conditions; the respective methodology will also be presented here.

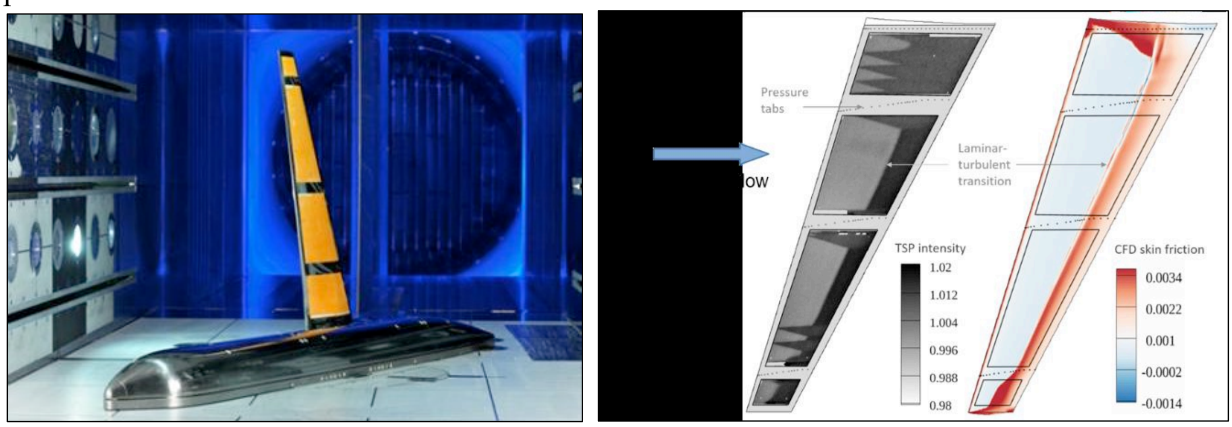


Fig. 1: *NLF-FSW wing body model in ETW and measured and predicted transition line*

References

[1] A. Seitz, A. Hübner and K. Risse. The DLR TuLam project: design of a short and medium range transport aircraft with forward swept NLF wing. CEAS Aeronautical Journal 11, 449–459 (2020). https://doi.org/10.1007/s13272-019-00421-1

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 234-2

Aerodynamic Design of a High-Aspect Ratio Natural Laminar Wing for a New Short and Medium Range Aircraft

Javier Ruberte-Bailo[1], Alexander Büscher[2], Thomas Streit[3], Geza Schrauf[4,*]

[1] DLR, Lilienthalplatz 7, 38108 Braunschweig, Germany, javier.rubertebailo@dlr.de
[2] Airbus, Airbus Allee 1, 28199 Bremen, Germany, alexander.buescher@airbus.com
[3] DLR, Lilienthalplatz 7, 38108 Braunschweig, Germany, th.streit@dlr.de
[4] Retired from DLR, 28203 Bremen, Germany, contact@schrauf.de

Key Words: *Drag Reduction, Friction Drag, Transition, Laminar Flow, CATNLF*

Improving the aerodynamic performance of a transportation aircraft is an important contributor to achieve the ambitious goal to attain zero net emission in this transportation sector. The reduction of wing friction drag by delaying transition to keep flow laminar is one of the most promising efforts to succeed here.

The oral presentation will show the aerodynamic design and analysis of a highly efficient high aspect ratio wing using natural laminar flow (NLF) shape design intended for a new short and medium range aircraft.

Unlike previous NLF designs, the cruise design speed is fully comparable to turbulent wing design in this aircraft category. Nevertheless, the excellent aerodynamic performance of the wing is based on both the vortex drag benefit achieved by the span increase but also the large laminar extent of the wing flow. This is realised by using CATNLF [1, 2] section design.

The oral presentation will show the complete design loop from starting with given top level aircraft requirements (TLAR) and deduced aero requirements, initial section design and wing planform studies and resulting in the final section design to maximise the overall aero performance benefit considering all drag components for the entire aircraft mission.

The entire design study is based on 2.5D, 2.75D and finally fully inverse 3D CFD simulations. For transition prediction a linear stability e^N method [3] was used, which provides N-factors for Tollmien-Schlichting and cross-flow instabilities. During the investigation wing sweep and corresponding aircraft cruise speed were increased step-by-step to keep the achieved laminar extent for a robust wing design at design and also off-design conditions.

References:

[1] Campbell, R. L. and Lynde, M. N., "Natural Laminar Flow Design for Wings with Moderate Sweep", 34th AIAA Applied Aerodynamic Conference, AIAA AVIATION Forum, AIAA 2016-4326, June 2016., https://doi.org/10.2514/6.2016-4326.

[2] Streit, T., Seitz, A., Hein, S., Kunze, P., "NLF Potential of Laminar Transonic Long-range Aircraft", AIAA Aviation 2020 Forum, 15-19. June 2020, USA Virtual Event, doi: 10.2514/6.2020-2748.

[3] Schrauf, G., "LILO 2.1 – Users Guide and Tutorial", GSSC Technical Report, July 2006.G. Schrauf: "Status and perspectives of laminar flow." The Aeronautical Journal, December 2005, Vol. 109, No. 1102, pp. 639-644.

STS 234-3

Aernnova Laminar Technology Demonstrators

Miguel Á. Castillo Acero[1]* and Federico Martin De La Escalera Cutillas[2]

Aernnova, Madrid, Spain, https://www.aernnova.com
[1]VP Technology Development, miguelangel.castillo@aernnova.com
[2] Head of Research & Technology, federico.martindelaescalera@aernnova.com

Key Words: *Drag Reduction, Friction Drag, Laminar Flow, Hybrid Laminar Flow Control*

The 7th Framework Programme for Research and Technological Development of the European Union (EU) included Clean Sky Joint Undertaking (JU) as a public-private initiative. The Clean Sky goal was to develop new technologies to achieve more environmentally friendly aircraft, carrying out all the steps to reduce the environmental impact of commercial transport aircraft. This was translated into integrating, demonstrating, and validating technologies capable of reducing CO2, NOx and noise emissions between 20 to 30% compared to the benchmark aircraft of 2010.

Clean Sky 2 (CS2) has been the successor programme within Horizon 2020 - the EU's research and innovation funding programme and has had an additional objective to those of the original Clean Sky: to keep on develop new technologies for reducing environmental footprint and a strong aeronautical industry and supply chain in Europe to compete globally.

Aernnova has been a partner of Clean Sky and CS2 during the last fourteen years and have dedicated numerous resources for the development, design, manufacture, and validation of laminar wing like type surfaces. The challenges have included natural laminar flow and hybrid laminar flow control technology demonstrators. Aernnova, together with partners, have integrated new solutions in ground and fly demonstrators to answer the technological challenges and a further step in its commitment to play a leading role towards a low-emission and competitive aeronautical industry. This contribution summarizes these developments and demonstrators.

References:

[1] G. Schrauf: "Status and perspectives of laminar flow." The Aeronautical Journal, December 2005, Vol. 109, No. 1102, pp. 639-644.

[2] G. Williams: "Aerospace Technology Demonstration: BLADE, the Flagship Laminar Flow Project within the Clean Sky Programme," SAE Technical Paper 2017-01-2016, 2017, https://doi.org/10.4271/2017-01-2016.

[3] M. A Castillo Acero: "Innovating with High Precision Tools," Innovation Takes Off - Clean Sky, European Research for Aeronautics. Le Cherche Midi. 2016

[4] Clean Aviation; "HLFC on Tails & Wing." https://clean-aviation.eu/clean-sky-2/demonstrators/key-demonstrators/hlfc-on-tails-wing.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 234-4

Multidisciplinary Optimization of Load Adaptive Wings for Highly Efficient Long-Haul Airliners

Tobias F. Wunderlich[1,*] and Sascha Dähne[2]

[1] German Aerospace Center, Institute of Aerodynamics and Flow Technology, 38108 Braunschweig, Lilienthalplatz 7, Germany, tobias.wunderlich@dlr.de

[2] German Aerospace Center, Institute of Lightweight Systems, 38108 Braunschweig, Lilienthalplatz 7, Germany, sascha.daehne@dlr.de

Key Words: *Multidisciplinary Design Optimization (MDO), adaptive wing, manoeuvre load alleviation (MLA), fiber composite, structural sizing, fluid-structure coupling*

As part of the DLR project oLAF (optimal load-adaptive aircraft), a long-haul airliner with maneuver load reduction limited to aileron deflections and the use of a geared turbofan engine is being designed and optimized. Adaptive wing technologies based on trailing edge control surface deflections to reduce drag at cruise and for optimal load reduction are introduced and supplemented by other structural technologies with increased strain allowable to reduce wing mass. Using multidisciplinary design optimization, an aircraft design using these technologies has been made and is compared to the reference aircraft.

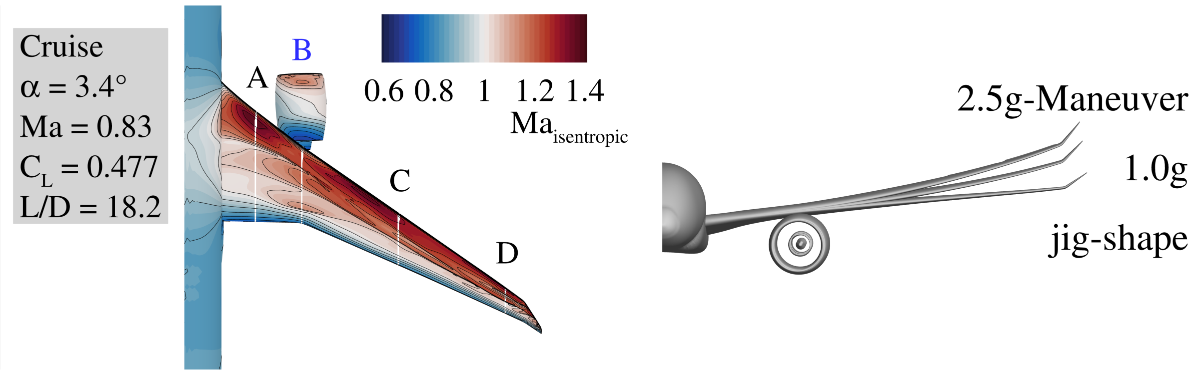


Figure 1: *Isentropic Mach number distribution and wing deformations of the reference aircraft.*

In this work the results of the aero-structural wing optimizations of the reference aircraft and the aircraft with a load adaptive wing will be presented. In this optimization process [1], high-fidelity simulation methods are used to determine the flight performance in the transonic cruise flight, the loads of the wing in maneuver flight and the mass of the wing box made of fiber composite materials. Static aeroelastic effects are considered in all flight conditions. The minimization of the fuel consumption for three typical flight missions represents the objective function. The geometric integration of the landing gear and the control surfaces, the tailplane sizing and aircraft trimming are considered. The selected design parameters describe the twist distribution and the control surface deflections. The results include a detailed presentation of the aerodynamic performances in the cruise and manoeuvre flight, the wing deformations and the structural mechanical properties after structural sizing (see examples in Figure 1).

References

[1] T. F. Wunderlich et al. "Global Aerostructural Design Optimization of More Flexible Wings for Commercial Aircraft". In: Journal of Aircraft 58.6 (2021), pp. 1254–1271. doi: 10.2514/1.C036301.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 234-5

Fuel Burn Efficiency Potential of Load Alleviation and Wing Planform Optimization in Conceptual Overall Aircraft Design

Markus Dino Krengel[1,*]

[1] Research Engineer, Institute of Aerodynamics and Flow Technology, German Aerospace Center, Lilienthalplatz 7, 38108 Braunschweig, Germany, markus.krengel@dlr.de

Key Words: *Aircraft Design, Load Alleviation, Aeroelasticity, MDO, Span Extension*

Load alleviation as a technology can increase the fuel efficiency of an aircraft design by reducing the wing mass. Furthermore, the reduction of aerodynamic loads on the transonic, swept outer wing allows for higher aspect ratio wings. Load alleviation therefore is important for the aircraft design and should be considered within the conceptual design phase.

Here, a design process is shown that integrates the relevant disciplines into the overall aircraft design through simplified but physics-based models based on the MIT tool ASWING [1]. The over the air download (OAD) integration tool is OpenAD [1] from DLR. Surrogate model-based optimizations of the wing shape are done with respect to the fuel consumption. The design space contains nine geometrical wing shape parameters. Building upon the research of Krengel and Hepperle [2], the findings reveal that active load alleviation reduces the wing mass. The effect of load alleviation varies with the wing span. Optimization results show a fuel burn reduction potential of up to 11.6% compared to an optimized baseline design. The overall aircraft impact of load alleviation is also discussed in detail. This includes an exploration of the effects of simplified boundary conditions on the design. Moreover, the influence of various design parameters on the aircraft design, both with and without load alleviation, is assessed in depth based on neuronal network surrogate models.

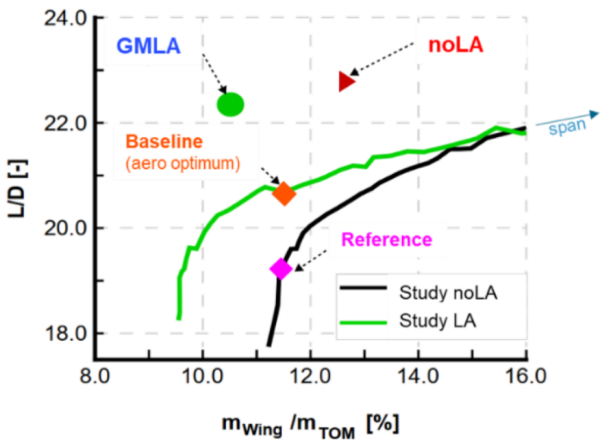


Figure 1: *Lift-over-drag over the relative wing mass for selected span studies and the found*

References:

[1] Drela, M., Integrated simulation model for preliminary aerodynamic, structural, and control-law design of aircraft, 40th Structures, structural dynamics, and materials conference and exhibit, p. 1394, doi: 10.2514/6.1999-1394

[2] Krengel, M., Hepperle, M, Gust and Maneuver Load Alleviation in Conceptual Aircraft Design, STAB/DGLR Symposium. Cham: Springer Nature Switzerland, 2022, doi: 10.1007/978-3-031-40482-5. ISBN 9783031404818.

[3] Woehler, S., Atanasov, G., Silberhorn, D., Fröhler, B., and Zill, T., Preliminary Aircraft Design within a Multidisciplinary and Multifidelity Design Environment, Aerospace Europe Conference 2020, Bordeaux, France, 2020.

STS 237

Multi-Disciplinary Design and Optimisation of Novel HEX (Heat-Exchangers) for Green Aviation

Chair: Shahrokh Shahpar

Innovation Hub, Future Methods, Rolls-Royce plc, Derby, DE24 8BJ, United Kingdom
Shahrokh.Shahpar@Rolls-Royce.com

Session Abstract

Key Words: *Topology Optimisation, Multi-physics simulation, ALM, Hydrogen fuel cells, AI-Machine-learning, Multi-fidelity design and Robust Design.*

With an increased interest in electrification for aviation, attention is drawn to optimising the overall aircraft/engine thermal management, which involves designing more efficient heat exchangers (HEX). In this STS, various technologies and software available for both design and optimisation of cold-plate HEXs will be discussed and reviewed. The focus would be on methods that can both analyse the heat sources that represents power electronics components for aviation applications and also design new novel geometries to address the cooling requirements.

One of the key technologies we would like to discuss is the thermal-fluid topology optimisation [1] for HEXs for turbomachinery jet-engine applications. Some of the Key technologies for an optimum design of HEXs would be hi-fidelity conjugate CFD simulation, automatic meshing of very complex geometries, multi-physics simulation, advances in Additive-layering Manufacturing (ALM) as well as testing and experimentation to support the design. Novel shapes resulting from topology optimisation that could only be manufactured using ALM will also be exhibited and discussed. Furthermore, we explore how these technologies can be extended to other design problem like the optimum fuel passages for PEMFCs (Hydrogen Fuel Cells), etc.

Presentations are also expected from an ongoing R&D work as part of the EU Horizon Program (NextAir) to produce a fully-fledged Digital Twin of a Heat-Exchanger [2].

References:

[1] Raske, N, Ausin Gonzalez, O, Furino, S, Pietropaoli, M, Shahpar, S, & Montomoli, F. "Thermal Management for Electrification in Aircraft Engines: Optimization of Coolant System." Proceedings of the ASME Turbo Expo 2022, Netherlands. June, 2022. V06BT13A013. ASME. https://doi.org/10.1115/GT2022-82538

[2] NEXTAIR -multi-disciplinary digital -enablers for NEXT-generation AIRcraft design and operations, (**WP6:** Digital Twin of a HEX) -DOI: 10.3030/101056732, Sept 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 237-1

Multi-disciplinary Optimization of Gyroid Structures for a Cold Plate Heat Exchanger Design

Elhusseiny Daifalla[1], Mario Carta[2], Shahrokh Shahpar[3]*, and Indi Tristanto[4]

[1] Undergraduate Researcher (Intern), Innovation Hub, Future Methods,
Rolls-Royce plc, Derby, DE24 8BJ, UK
edaifalla@gmail.com

[3] Rolls-Royce Fellow, Innovation Hub, Future Methods,
Rolls-Royce plc, Derby, DE24 8BJ, UK
Shahrokh.Shahpar@Rolls-Royce.com

[2] Postdoctoral Researcher, Department of Mechanical, Chemical and Materials Engineering,
University of Cagliari, 09123 Cagliari, Italy
mario.carta@unica.it

[4] CFD Developer, Innovation Hub, Future Methods,
Rolls-Royce plc, Derby, DE24 8BJ, UK
indi.tristanto@rolls-royce.com

Key Words: *High performance heat exchanger, multi-disciplinary optimization, simulation*

The strive for electrification in today's civil aviation industry has brought the need for increasingly powerful on-board electrical systems, including those meant to replace heavier, bulkier hydraulic flight control devices. These high-performance electrical components are bound to produce large amounts of heat that, if not dissipated properly, will lead to malfunctioning and even permanent damage. For this reason, high performance heat exchangers represent a key enabler for future advances in aircraft systems electrification and are vital to meet net zero goals and reduce our aviation's carbon footprint. For a given volume of the exchanger, the heat flow rate can be increased by adopting more sophisticated fluid domains. However, excessive geometrical complexity will lead to an increase in pressure losses, often resulting in inhomogeneous temperature distributions. I

n this paper, a novel optimization procedure is employed to maximize the efficiency of a high-performance heat exchanger, while minimizing overall pressure loss and temperature gradients. The optimization is performed with full-3D high-fidelity computational flow simulations using the commercial code Ansys Fluent. The geometry of the fluid domain is constituted by triply periodic minimal surfaces (TPMS), with a parametrization based on thickness and aspect ratios, done by using the n-Topology suite. To assess the performance gain, the optimized TPMS design is compared against the baseline one and a conventional serpentine design.

Acknowledgements
The information in this document is proprietary and confidential to Rolls-Royce and is available to authorised recipients only – copying and onward distribution is prohibited other than for the purpose for which it was made available.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 237-2

Multidisciplinary Optimisation of Additive Manufactured Heat Exchangers for Aeronautical Applications

Alessandro Chiodi[1], Alessandro Alaia[2,*], Eduardo Lombardi[2], Konstantinos Gkaragkounis[2], and Shahrokh Shahpar[3]

[1] Politecnico di Torino, 10129 Turin, Italy, alessandro.chiodi@polito.it

[2] Optimad Engineering srl, Turin, Italy, alessandro.alaia@optimad.it

[3] Innovation Hub, Future Methods, Rolls-Royce Plc, Derby DE24 8JY, UK, shahrokh.shahpar@Rolls-Royce.com

Key Words: *Generative Design, Aerothermal Topology Optimisation, Multiscale Optimisation, Reduced Order Modelling, Novel Heat Exchangers*

Computational process of *Topology Optimization* (TO) aims to determine location, shape, and connectivity of voids within a solid design space to maximize the value of an objective function, under specified constraints. In the context of heat exchangers, TO offers enhanced design flexibility compared to conventional shape optimization, yielding improved performance and reduced weight. *Additive Manufacturing* (AM) is emerging as a versatile solution for crafting intricate geometries resulting from TO, overcoming the limitations of traditional manufacturing processes.

The objective of this research is to establish a proof-of-concept workflow for the TO of additive manufactured MCHX. The envisioned application is a cutting-edge cooling system for electric flight, where MCHX serves as a pivotal component for managing the working temperature of batteries, fuel cells, and preventing overheating of electronic equipment. The test case involves a MCHX for on-board power electronics. Optimization goals include maximizing the heat rejection, while keeping the compliance to constraints on maximum temperature on the solid, and pressure drop across the device.

TO of MCHX is intrinsically a multi-scale problem. Performances at the macroscale depend on both the flow regime at the scale of channels and lattice geometry. Simultaneously, microscale phenomena are driven by operating conditions at the macroscale. To address this, homogenization theory is employed, treating the lattice matrix as an equivalent porous medium, whose permeability and heat transfer coefficients are considered non-linear functions of flow conditions and local lattice geometry. A multi-fidelity *Machine Learning* (ML) model is trained offline with data coming from *Direct Numerical Simulations* (DNS) on a wide set of lattice cells (microscale) to provide closure relationships for permeability, Forchheimer's, and heat transfer coefficient. This approach has the two-fold advantage of reducing the overall cost associated to the optimization and improving robustness. CFD simulations at the microscale are performed in OpenFOAM, while Immerflow (Optimad) has been employed for macroscale simulations. The optimization process is managed by modeFRONTIER.

The resultant optimized geometries exhibit an uneven distribution of lattice parameters, leading to a substantial increase in flow path tortuosity for the coolant. The increased tortuosity leads to an increase in the residence time of the coolant which has a beneficial effect on heat dissipation as demonstrated by the 27% increase in heat rejection compared to a uniform lattice.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 237-3

Aerothermal Topology Optimisation of a Compact Heat Exchanger

Shahrokh Shahpar[1]

[1] Rolls-Royce Fellow, Innovation Hub, Future Methods,
Rolls-Royce plc, Derby, DE24 8BJ, United Kingdom
Shahrokh.Shahpar@Rolls-Royce.com

Key Words: *Aerothermal topology optimisation, Compact heat exchanger, high-fidelity CFD, RANS, grid-independent solution*

Future aircraft power electronics requires careful management of the heat generated by the resistors. This involves designing and optimising more efficient compact heat exchangers (HEX). In this paper, a cold plate HEX will be defined in detail such that other researchers can make use of it for their future comparative and validation studies. High-fidelity simulation results for the traditional, namely serpentine cooling passages will be provided and compared to different topology optimised shapes.

In particular, a class of triply periodic minimal Surfaces (TPMS) namely *Gyroids* and *Diamonds* are used as initial shapes for the fluid passages. The design space is made of the TPMS periods in three *x, y & z* directions, and their relative thickness variations. The objective functions defined as the peak surface temperature, gradient of the temperature and total pressure, all of which are aimed to be minimised. The coolant fluid is a mixture of water and ethylene-glycol mixture.

The performance of the HEX is assessed using a high-fidelity CFD model making use of conjugate heat transfer by solving the RANS equations with k-ω SST model in Ansys Fluent. Different fluid and solid meshes have been generated using ICEM and an in-house version of the Boxer, a cut-cell Cartesian meshing tool. A grid-independent solution was sought, aiming for a y+ of 1 on all viscous surfaces.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 237-4

Multidisciplinary Optimization of a Heat Exchanger Using Lattice Structures Accounting for Additive Manufacturing Constraints

Nicolas Roussouly[1,*], Francois Gallard[1], Ahmad Batikh[2], Jean-Pierre Fradin[2] and Guillaume Chaumont[3]

[1] IRT Saint-Exupéry, 3 Rue Tarfaya, 31400 Toulouse, France
nicolas.roussouly@irt-saintexupery.com, francois.gallard@irt-saintexupery.com

[2] Icam - Institut Catholique d'Arts et Métiers, 75 Av. de Grande Bretagne 31300 Toulouse, France
ahmad.batikh@icam.fr, jean-pierre.fradin@icam.fr

[3] Akkodis, 7 Bd Henri Ziegler 31700 Blagnac,
guillaume.chaumont@akkodis.com

Key Words: *Multidisciplinary Design Optimization, Additive Manufacturing, Lattice Structures, Surrogate Modelling, Gradient-Free Optimization, GEMSEO.*

In the scope of the RECET4Rail European Union (EU) project we aim at better understanding how Additive Manufacturing (AM) technologies can contribute to Traction Drive (TD) sub-systems through the development and manufacturing of new types of heat exchangers based on lattice structures. In particular, we focus on the design of an AM-based heat exchanger through a Multidisciplinary Design Optimization (MDO) process. We show how the GEMSEO framework [1] (https://gemseo.readthedocs.io) can be used with several features (sensitivity analysis, surrogate modeling, gradient-free optimization, etc) to implement the optimization process from industrial tools despite the complexity of the problem and accounting for several practical constraints.

The process is based on three chained tools which are wrapped into GEMSEO disciplines: (1) a CAD builder that parametrizes the geometry, (2) an automatic mesher based on ANSYS Design Modeler and Meshing, (3) a coupled fluid-thermal model, implemented through the ANSYS Fluent solver. Each tool is installed on a different execution machine with a different operating system. Especially, the coupled fluid-thermal model runs on a distant HPC through a job scheduler access. Platform services from GEMSEO are used to handle those distributed executions thanks to ssh-based disciplines and scheduler based disciplines. The optimization problem is formulated in order to minimize the thermal resistance of the heat exchanger subject to an allowable pressure loss. The design variables characterize the shape of the lattice structures such as radii, orientations, cell density which potentially leads to a huge dimension.

Manufacturing constraints have been taken into account through the range of design variables in order to ensure minimal diameters, limited overhanging and free path for powder removal. In addition, the outputs of the chain are not differentiated with respect to the inputs which prevents the use of gradient based algorithm. Those difficulties are limited by reducing the number of design variables through control points values interpolation, and separating the global analysis into three phases: (1) an initial Design Of Experiment (DOE) with a sensitivity analysis, (2) an optimization based on surrogate models built from the initial DOE, (3) a gradient-free optimization on the real model. Through this analysis, we show how GEMSEO can be conveniently manipulated to converge toward an optimal solution that satisfies the constraints and enables to gain 28% on the thermal resistance with a pressure loss increase of 42%, while satisfying the allowable pressure loss.

References

[1] Gallard, F., Vanaret, C., Guénot, D., Gachelin, V., Lafage, R., Pauwels, B., Barjhoux, P.-J., and Gazaix, A., "GEMS: A Python Library for Automation of Multidisciplinary Design Optimization Process Generation," *2018 AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, 2018, p. 0657

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 237-5

Towards Dynamic Modelling of Thermal Management Systems for Greener Aviation

Indi Tristanto[1]*, Aidan Burns[2], Paolo Malgieri[3] and Jason Chetwynd-Chatwin[4]

[1] Future Methods, Rolls-Royce Plc, Derby, DE24 8BJ, UK, indi.tristanto@rolls-royce.com

[2] Future Methods, Rolls-Royce Plc, Derby, DE24 8BJ, UK, aidan.burns@rolls-royce.com

[3] Thermal and Fluid Systems, Rolls-Royce Plc, Bristol, BS34 7QE, UK, paolo.malgieri@roll-royce.com

[4] Thermal and Fluid Systems, Rolls-Royce Plc, Bristol, BS34 7QE, UK, jason.chetwynd-chatwin.se@rollsroyce.com

Key Words: *Thermo-fluid design analysis, effectiveness-NTU, heat exchanger, Modelica*

Greener aviation poses significant thermal management challenges. The shift towards greener aviation concepts and the consequent use of high-power electrical systems (for propulsion or as secondary systems) is likely to cause increased heat generation and the proliferation of low-grade heat sources. Similarly, the introduction of geared architecture technologies increases significantly the heat generated by the propulsion system compared to traditional architectures. Therefore, the capability to design, analyse and optimise thermal architectures becomes of great importance.

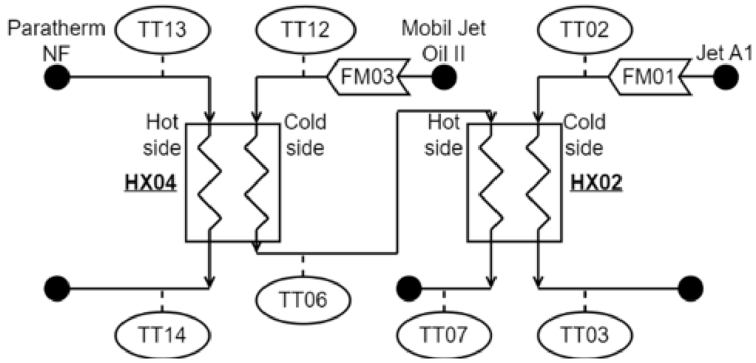


Figure 1: *1 a simple thermal network with three fluids and two heat exchangers*

Figure 1 shows a simple thermal network with multiple fluid loops and heat exchangers. Many methods of simulation currently rely on a steady-state approach based on effectiveness-NTU methods to analyse heat exchangers [1]. However, in future thermal architectures transient operations will be of great interest. Hence, a dynamic modelling capability for thermal management systems is needed.

Modelica is identified as a promising tool for this type of analysis [2], which will be extended for multi-disciplinary system design. This work presents an assessment of the capability of steady state modelling techniques as a key enabler.

Two heat exchanger models were considered for the HX02 and HX04 of Figure 1. Figure 2 shows the Modelica set up for the dynamic simulation of the heat exchangers. For both models two different steady state formulations were used for the overall heat transfer coefficients (U): formulation combination C uses a typical textbook value [3]; formulation combination D is a constant value derived from averaging the overall heat transfer coefficient (U) derived from the heat exchanger supplier data for HX02 and HX04 performance. Transient rig data was available for the thermal network of Figure 1 in terms of thermocouple and flow meter measurements (see ident locations of Figure 1).

The results of the Modelica analysis are compared with the rig temperature data in Figure 3 and Figure 4: as expected, C is performing relatively well during steady state operation but shows a discrepancy up to 5% in transient operation. While D is qualitatively satisfactory, it exhibit quantitative discrepancy even in the steady state operation.

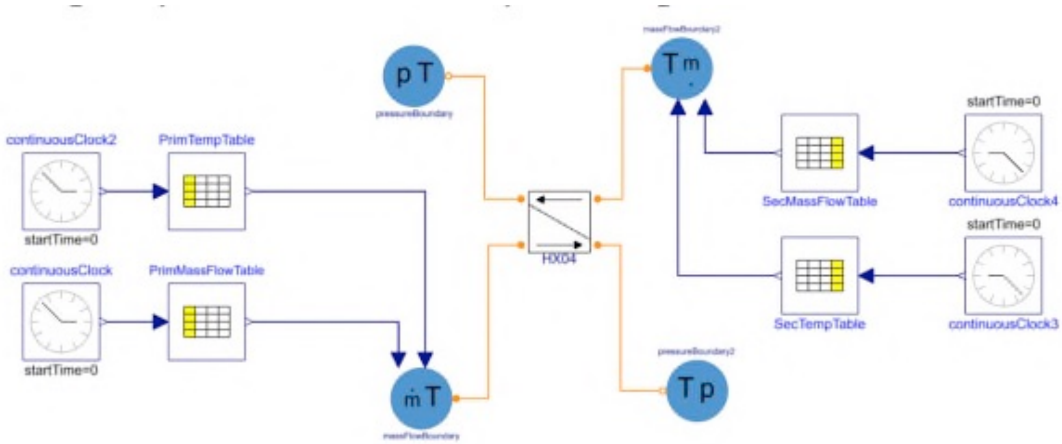


Figure 2: *Modelica set up for heat exchanger simulation*

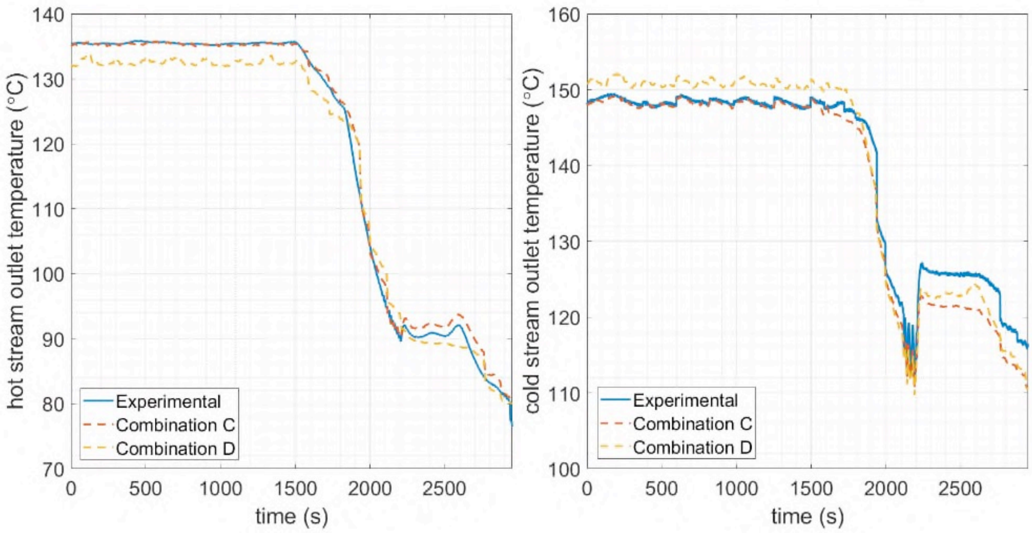


Figure 3: *HX04 cold and hot streams outlet temperatures*

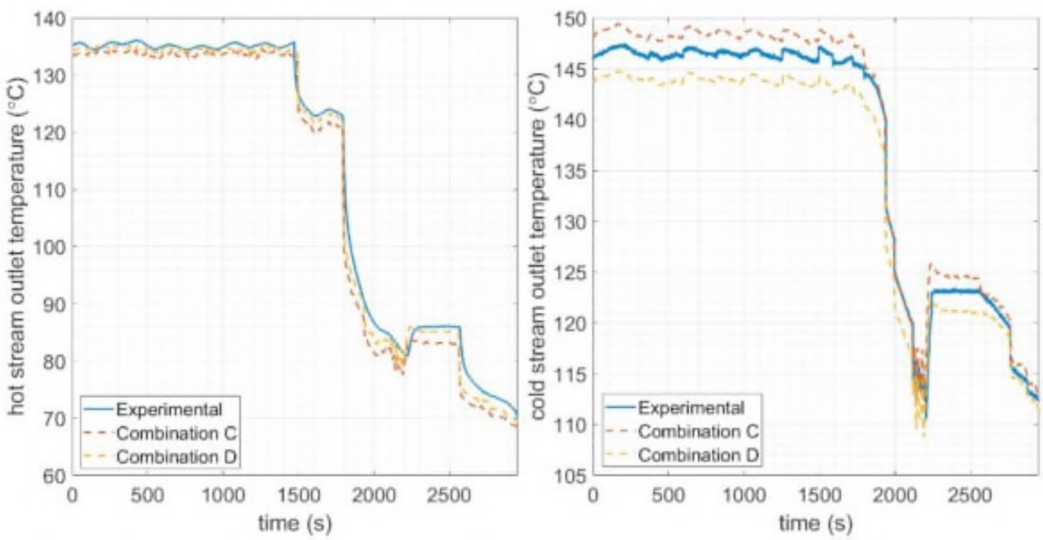


Figure 4: *HX02 cold and hot stream temperatures*

References

[1] J. Lienhard IV and J. Lienhard V, A Heat Transfer Textbook, Phlogiston Press, 2020.

[2] P. Fritzson, Introduction to Modeling and Simulation of Technical and Physical Systems with Modelica, John Wiley & Sons, Inc, 2011.

[3] J. Coulson and J. Richardson, Chemical Engineering, Volume 6, Pergamon Press, 1993.

9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS 2024
June 3-7, 2024, Lisbon, Portugal

STS 239

Cold Transportation

Pekka Neittaanmäki[1]

[1] Lappeenranta University of Technology Yliopistonkatu 34, 53850 Lappeenranta, Finland
Pekka.neittaanmaki@jyu.fi

Session Abstract

Key Words: *Transport, Cold Chain, Optimization, Artificial Intelligence*

Efficient transportation of perishable goods, such as food and medicine, poses substantial challenges in maintaining their quality and timely delivery. This session aims to address these hurdles by introducing innovative computational and technological solutions.
In this session, we will delve into enhanced logistics planning strategies that leverage cutting edge machine learning methods. By harnessing predictive analytics and adaptive algorithms, we aim to develop transport route planning, minimizing transit times and mitigating environmental impact.

The session will showcase a diverse range of research endeavors and practical implementations, emphasizing the crucial role of technology in elevating the standards of transporting perishable goods. Join us as we explore pioneering advancements in transportation technology and logistics management, setting the stage for more sustainable and efficient delivery systems for essential goods.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 239-1

The Role of Innovation in Adapting to Legal and Regulatory Changes as Europe Goes Green

Isla Usvakangas[1,*], Ronja Tuovinen[2], Pekka Neittaanmäki[3]

[1] University of Helsinki, PL 4 (Yliopistonkatu 3) 00014 University of Helsinki, Finland, isla.usvakangas@gmail.com
[2] Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, ronja.tuovinen@gmail.com
[3] Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, pekka.neittaanmaki@jyu.com

Key Words: *Cold chain, cold transportation, CO2 emissions, environmental goals, AI*

Due to the rising awareness of the climate crisis different legislative bodies have revised and added to laws and regulations around emissions and environmental protections. This forces companies and organizations to upgrade and evolve when it comes to green technologies. New innovations using algorithms and AI provide invaluable tools in achieving this shift. In this presentation I will discuss, in the context of logistics and cold transport, the current and changing legal landscape around emissions, phasing out fossil fuels, and natural protection, and what this means for the future of the field.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 239-2

Cold Transportation Systems and their Impact on the Environment

Ronja Tuovinen[1,*], Veronika Akimova[2], Kirill Akimov[3] and Pekka Neittaanmäki[4]

[1] Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, ronja.tuovinen@gmail.com

[2] University of Helsinki, PL 4 (Yliopistonkatu 3) 00014 University of Helsinki, Finland, veronika.akimova@helsinki.fi

[3] University of Helsinki, PL 4 (Yliopistonkatu 3) 00014 University of Helsinki, Finland, kirill.akimov@helsinki.fi

[4] Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland.

Key Words: *Cold chain, Environmental impact, Optimization, Vehicle routing problem*

Cold transportation and its impact on the environment is one of the biggest challenges that we are facing. This problem has many sides and many different solutions to it. The biggest obstacles are faced with maintaining the cold chain, rising CO2 emissions, and inefficient transportation and cooling methods. The most promising solutions are optimization, the use of AI methods, and more efficient solutions for maintaining the cold chain. In this presentation we will discuss the refrigerated transportation system, how it impacts the environment, and what are the weak points we should consider.

STS 239-3

Optimisation of Temperature-Controlled Transportation

Kirill Akimov[1,*], Pekka Neittaanmäki[2], Veronika Akimova[3], and Ronja Tuovinen[4]

[1] University of Helsinki, P.O. Box 4 (Yliopistonkatu 3), 00014 University of Helsinki, Finland, veronika.akimova@helsinki.fi

[2]Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, pekka.neittaanmaki@jyu.fi

University of Helsinki, P.O. Box 4 (Yliopistonkatu 3), 00014 University of Helsinki, Finland, kirill.akimov@helsinki.fi

[4]Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, ronja.tuovinen@gmail.com

Key Words: *artificial intelligence (AI), machine learning (ML), temperature-controlled transportation, cold chain logistics*

We identify potential areas where artificial intelligence (AI) may be applied to optimise the cold chain logistics. The advantages of AI over human beings in the operation of refrigerated trucks and management of supply chains are overviewed. In particular, the potential of machine learning for implementation of self-driving trucks is considered, while the potential of predictive artificial intelligence for optimal product distribution is noted. Moreover, we identify the risks arising from the implementation of artificial intelligence.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 239-4

Artificial Intelligence in Temperature-Controlled Transportation

Veronika Akimova[1]*, Pekka Neittaanmäki[2], Kirill Akimov[3], and Ronja Tuovinen[4]

[1] University of Helsinki, P.O. Box 4 (Yliopistonkatu 3), 00014 University of Helsinki, Finland, veronika.akimova@helsinki.fi

[2]Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, pekka.neittaanmaki@jyu.fi

[3]University of Helsinki, P.O. Box 4 (Yliopistonkatu 3), 00014 University of Helsinki, Finland, kirill.akimov@helsinki.fi

[4]Lappeenranta University of Technology, Yliopistonkatu 34, 53850 Lappeenranta, Finland, ronja.tuovinen@gmail.com

Key Words: *artificial intelligence (AI), machine learning (ML), temperature-controlled transportation, cold chain logistics*

We identify potential areas where artificial intelligence (AI) may be applied to optimise the cold chain logistics. The advantages of AI over human beings in the operation of refrigerated trucks and management of supply chains are overviewed. In particular, the potential of machine learning for implementation of self-driving trucks is considered, while the potential of predictive artificial intelligence for optimal product distribution is noted. Moreover, we identify the risks arising from the implementation of artificial intelligence.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 239-5

Passive Cold Transportation Method

Juha Harjula[1], Jukka Proskin[2]

[1] CHS Group, Turvalaaksonkuja 2, 01740 Vantaa, Finland, juha.harjula@chs.fi

[2]Coldest Chain oy, Kauppakatu 28 A 59, 40100 Jyväskylä, Finland, jukka.proskin@gmail.com

Key Words: *Cold chain, passive cooling, cold transportation*

Passive refrigerated transport unit:

1) Saves CO2 in emissions and expenses due to top class insulation, which does not require external energy to sustain the wanted temperature.

2) There is no need for expensive and rarer special equipment. Any unit or vehicle can be used, so loading and filling rate can be optimized and maximized. This reduces CO2 and saves costs.

3) The transport chain is stable and safe because there is no need for external supply of energy during the trip. This eliminates the risk of breaking the cold chain due to running out of energy or breaking of vehicle. This reduces waste and spoilage.

4) The transportation unit is recyclable and useable in multiple purposes. The unit can be directly placed where the customer needs it, and separate sales units are not needed. This saves materials, costs, and CO2.

STS 240 A & B

Advanced CFD Applications for Complex Aircraft Configurations

Chair: Jochen Wild[1]

[1] DLR German Aerospace Center, Institute of Aerodynamics and Flow Technology, Lilienthalplatz 7, 38108 Braunschweig, Germany, Jochen.Wild@dlr.de, https://www.dlr.de/as

Session Abstract

Key Words: *Applied aerodynamics, computational fluid dynamics (CFD), high-lift aerodynamics, flow control, load alleviation, propulsion*

The applied aerodynamics analysis of transport aircraft still poses high challenges on the capabilities of numerical simulations. In past years, this series of STS focused specifically on the high-lift regime, where the demand on accurately predicting stall onset achieved to a sophisticating level. Nowadays, new challenges arise with the progress on active flow control technologies and load control as well as new types of propulsion. This issue of the STS is intended to provide insight into such activities tackling the improvement of simulation capabilities for complex aircraft configurations, as there are:

- low-speed assessment of different propulsion concepts;
- unsteady active flow control for load alleviation;
- simulation of strut-braced wing aircraft.

Contributions are coming from running international research projects governed by EU funded Research and Innovation Actions, contributions to Clean Sky 2 and Clean Aviation Joint Undertakings, as well as other national and international cooperation activities. The following contributions are foreseen:

1. “Unsteady simulations of an aircraft in take-off conditions with installed engine and rotating fan” by F. Sartor, F. Moens & O. Atinault (ONERA)
2. “About the Ability of Anisotropic Mesh Adaptation to Capture Complex Physics on a Transonic Tandem Compressor” by E. Guilbert*, A. Remigi, M. Philit & F. Alauzet (Equipe Inria-ONERA & Safran Tech)
3. “Numerical Analysis of a Large Scale Distributed Propulsion Experiment at High Lift” by J. Oldeweme, T. Lindner, C. Bode, P. Scholz & J. Friedrichs (TU Braunschweig)
4. Aerodynamic Investigation of a Propeller-Driven Transport Aircraft with Distributed Propulsion within the IMOTHEP Project” by D. Keller, A. Visingardi, L. Wiart, Y. Maldonado, F. Morlando & G. Andreutti (DLR, CIRA & Safran Tech)
5. “Increasing Take-Off Performance of a Distributed Propulsion Wing Section using Segmented Flaps” by T. Lindner, J. Oldeweme, J. Friedrichs & P. Scholz (TU Braunschweig)
6. “Investigating Truss-Braced Wing Configuration through CFD Based Analysis” by M. Hothazie, D. Crunteanu, I. Bunescu & M. Pricop (INCAS)
7. “Fluidic Actuation for Gust Load Reduction on an Aircraft Wing” by A. Bauknecht, F. Siebert& K. Khalil (TU Braunschweig)
8. “Assessment of engine/airframe aerodynamic performance: comparison between ONERA, DLR, NLR and Airbus” by F. Sartor, A. Stürmer, M. Laban, S.R. Janssen & B. Caruelle (ONERA)

Contributing papers to the session parts A and B:
Part A: STS 240-1 to STS 240-4; **Part B:** STS 245-5 to STS 245-8;

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 240-1

Unsteady Simulations of an Aircraft in Take-Off Conditions with Installed Engine and Rotating Fan

Fulvio Sartor[1]*, Frédéric Moens[1] and Olivier Atinault[1]

[1] ONERA – The French Aerospace Lab, 8 rue des Vertugadins, 92190 Meudon, France,
fulvio.sartor@onera.fr

Key Words: *CFD, rotating fan, high-lift, take-off, installation effect, engine performance*

Numerical simulations have been performed to reproduce the flow around an aircraft in take-off conditions with installed engine, rotating fan and high-lift devices, by modelling the external aerodynamics of the plane as well as the internal aerodynamics in the secondary flow of the engine, located under the wing. These calculations take into account most of theelements of a transport aircraft and allow to finely model the interactions between theengine and the airframe. The analysis allows for an evaluation of both internal and external flow features of a civil aircraft equipped with engines with very high bypass ratio.

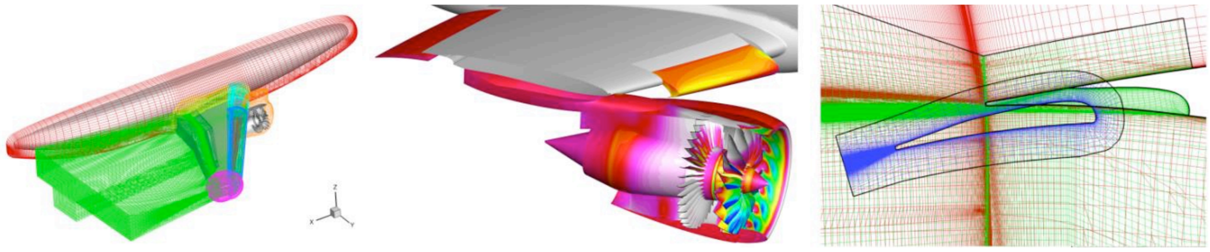


Figure 1: *Overview of the configuration (left), high-lift devices (center), Chimera assembly detail (right)*

The identified test case is the fusion between one of the NOVA configurations [1], the aircraft demonstrator designed by ONERA, with the ASPIRE engine [2], resulting from a previous collaboration between the partners [3]. This document presents the technical activities performed at ONERA on the NOVASPIRE configuration in take-off conditions with rotating fan, in the frame of the ADEC project, funded by the Clean Sky 2 Joint Undertaking under the European Union's Horizon 2020 research and innovation programme, with the Grant Agreement No 945583. The starting point of the simulations is the cruise conditions with rotating fan, thoroughly described in a previous publication [4]. High-lift devices are added to the clean configuration with Chimera approach, as illustrated in Figure 1.

The article will first present the strategy adopted, the grids generation and the approach for both computing and post-processing. Then the results of the unsteady CFD simulations are presented and discussed, including both the external flow around the aircraft and an analysis of the engine performance. A comparison between the results obtained on the isolated engine during a previous project [2,3] is also included.

References

[1] L. Wiart, O. Atinault, B. Paluch, D. Hue, R. Grenon. “Development of NOVA Aircraft Configurations for Large Engine Integration Studies”, *AIAA 2015-2254*

[2] A. Burlot, F. Sartor, M. Vergez, M. Méheut, R. Barrier. “Method comparison for fan performance in short intake nacelle”, *AIAA 2018-4204*

[3] M. Méheut, F. Sartor, M. Vergez, M. Laban, R. Schnell, A. Stuermer, G. Lefevre. “Assessment of Fan/Airframe aerodynamic performance using 360° uRANS computations: Code-to-Code comparison between ONERA, DLR, NLR and Airbus”, *AIAA 2019-0582*

[4] F. Sartor, “Unsteady simulations of an aircraft with installed engine and rotating fan”, *EUCASS2023-597*

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 240-2

About the Ability of Anisotropic Mesh Adaptation to Capture Complex Physics on a Transonic Tandem Compressor

Éloi Guilbert*[1], Alberto Remigi[2], Mickael Philit[2] and Frédéric Alauzet[1]

[1] GammaO, Equipe Inria-ONERA - Centre Inria de Saclay, Université Paris-Saclay, 91120 Palaiseau, France, frederic.alauzet@inria.fr, https://www.inria.fr
[2] Safran Tech, Modelling and Simulation, 78114 Magny-les-Hameaux, France, alberto.remigi@safrangroup.com, www.safran-group.com

Key Words: *Anisotropic Mesh Adaptation, Mixing plane, CFD, Turbomachinery*

The scope of this paper is to show the ability of metric-based anisotropic mesh adaptation to accurately capture flow features on a common complex test case: a transonic compressor with variable stator vane in tandem arrangement. The metric-based anisotropic mesh adaptation framework has been already applied successfully to an isolated compressor [1] and a film-cooled nozzle guide vane turbine [2]. Therefore, this technology seems quite appropriate for a transonic tandem compressor configuration thanks to its ability to increase locally the accuracy of the solution. In this work, the metric-based anisotropic mesh adaptation framework is applied to the 1.5 stages of the TUDa-GLR-Open Stage transonic axial compressor involving several rows interaction [4]. Several aspects have to be taken into account to extend mesh adaptation for such configurations.

First, multi-row machine is composed of several rows which may have large domain volume variation and which involve different physics. In that context, error estimate cannot be applied as it. We propose an extension of the interpolation error estimate in order to equi-distribute the error over all the rows whatever their size and the physics involved. This provides consistency in the mesh adaptation process.

Second, the rotor-stator interaction is modeled with mixing plane boundary condition technique that has been first introduced by Denton et al. [3]. It allows removing the dependency of the results on the relative position between the rotor and the stator. The idea is to average quantities along the pitch direction at the outlet of the row, those averaged values are transferred to the next row. And, vice versa. Associating mesh adaptation with mixing planes requires that the position and the distribution radial discretization must be automatic and consistent with the current adapted mesh. Therefore, after each mesh adaptation, a specific radial discretization for the mixing plane is built based on the current adapted mesh size. These radial discretizations are different on either side of the mixing plane as the adapted meshes do not match on either side.

References

[1] F. Alauzet, L. Frazza, and D. Papadogiannis. Periodic adjoints and anisotropic mesh adaptation in rotating frame for high-fidelity RANS turbomachinery applications. Journal of Computational Physics, 450:110814, February 2022.

[2] Frédéric Alauzet, L Billon, E Parente, M Philit, Alberto Remigi, and Dimitrios Papadogiannis. Evaluation of heat transfer performance of a film-cooled turbine vane using metric-based anisotropic mesh adaptation. National Harbor, MD & Online, 2023.

[3] J. D. Denton. An Improved Time-Marching Method for Turbomachinery Flow Calculation. Journal of Engineering for Power, 105(3):514–521, July 1983.

[4] Christian Kunkel, Jan Werner, Daniel Franke, Heinz-Peter Schiffer, Fabian Wartzek, Christoph Biela, and Christoph Starke. Introduction and Commissioning of the New Darmstadt Transonic Compressor Test Facility. American Society of Mechanical Engineers Digital Collection, November 2019.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 240-3

Numerical Analysis of a Large Scale Distributed Propulsion Experiment at High Lift

Jonas Oldeweme[1,*], Till K. Lindner[2], Christoph Bode[1], Peter Scholz[2] and Jens Friedrichs[1]

[1] Institute of Jet Propulsion and Turbomachinery, Technical University Braunschweig, Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany, j.oldeweme@ifas.tu-braunschweig.de

[2] Institute for Fluid Mechanics, Technical University Braunschweig, Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany, t.lindner@tu-braunschweig.de

Keywords: *Distributed Propulsion, Actuator Disc, High-Lift, uRANS*

Distributed propulsion configurations are a promising concept for future aircraft systems, promising a significant increase in overall aircraft efficiency and thus a reduction in CO_2 and other emissions. Hereby, distributed propulsion configurations offer an increased design space in terms of reliability, safety and weight distribution. However, not all interactions in such integration have been investigated or understood. Hence, it is not yet possible to reliably predict the aerodynamic effects of close wing coupling at the edge of the flight envelope on the basis of aerodynamic models and simulations. This is particularly true in the high-lift regime, where the flow over the wing tends to detach and exhibits highly non-linear aerodynamic effects. Furthermore, the unsteady propeller-propeller and propeller-wing interactions are a major challenge on top. As part of the Clean Sky 2 project CICLOP, aerodynamic propeller-wing interactions of such a configuration at high lift were investigated experimentally [1]. The wind tunnel model features a two element wing $c = 0.8$ m with three co-rotating propulsion units with a diameter of $D_P = 0.6$ m.

The focus of this work is a validation of the numerical simulation methods of different complexity of the propeller (Actuator Disc, steady-state RANS and unsteady RANS) used as standard in the design process on the basis of the experimental data [1]. The aim is to provide an evaluation of a well-founded and reliable qualitative and quantitative prediction of the distributed propulsion configurations. The focus of the evaluation is on design parameters that determine both positive and negative aerodynamic influences as well as the interaction of the propellers with each other, the effect of the wing on the propeller and the effect of the propeller wake on the wing. In addition to a periodic boundary environment, comparisons will also be made in wind tunnel environment. Here, besides the wind tunnel walls, the suspension of the drive trains and other geometry of the wind tunnel are included. As performed in the experiment, different relative propeller positions and thrust level are considered. Both, the wing and propeller performance are highly dependent on different positions [2].

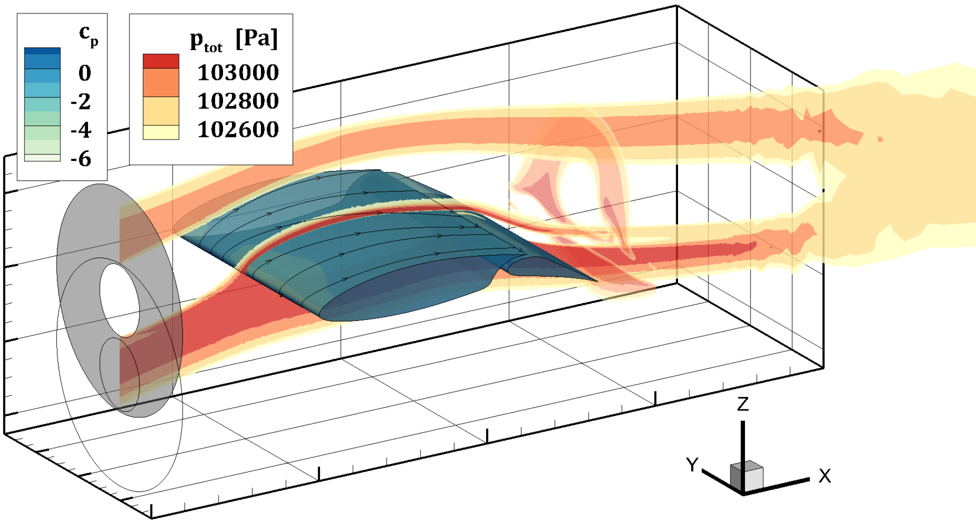


Figure 1: *Actuator disc RANS simulation of DP-model*

References

[1] Oldeweme J, Lindner T, Scholz P and Friedrichs J 2022 Experiment Design for a Distributed Propulsion Configuration at High Lift (Deutscher Luft- und Raumfahrtkongress (DLRK))

[2] Lindner T K, Oldeweme J, Scholz P and Friedrichs J 2023 Experimental Propeller Placement Analysis for a Distributed Propulsion Wing Section in High Lift Configuration (AIAA AVIATION 2023 Forum)

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 240-4

Aerodynamic Investigation of a Propeller-Driven Transport Aircraft with Distributed Propulsion within the IMOTHEP Project

Dennis Keller[1]*, Antonio Visingardi[2], Ludovic Wiart[3], Yebonne Maldonado[3,4], Fabrizio Morlando[2,5] and Giovanni Andreutti[2,6]

[1] German Aerospace Center (DLR), Lilienthalplatz 7, 38108 Braunschweig, Germany,
dennis.keller@dlr.de

[2] Centro Italiano Ricerche Aerospaziali (CIRA), Via Maiorise, 1, 81043 Capua (CE), Italy,
[2] A.Visingardi@cira.it, [5] fabrizio.morlando@cira.it, [6] G.Andreutti@cira.it

[3] Safran Tech, Rue des jeunes bois,78772 Magny-les-Hameaux, France,
[3] ludovic.wiart@safrangroup.com, [4] maldonado@safrangroup.com

Key Words: *Distributed propulsion, Aerodynamics, Propeller, High-Lift, CFD*

In the framework of the European IMOTHEP project, a novel regional propeller-driven transport aircraft concept with distributed propulsion is investigated [1]. A key feature of this aircraft design is the hybrid-electric approach allowing for fully electric flights on short range missions while extending the range with an integrated gas-turbine for longer range missions. Electrifying the powertrain allows to benefit from several advantages distributed propulsion promises such as reduced vertical tail size, reduced requirements for installed power, and potentially lower structural weight. Moreover, gains in overall aircraft efficiency may arise due to improved aero-propulsive efficiency.

The present publication illustrates the comprehensive collaborative work of Safran Tech, CIRA, and DLR that was carried out within the IMOTHEP project with regard to aero-propulsive efficiency of the investigated aircraft design. Various numerical studies on basic aspects like propeller position as well as detailed design studies on propellers, propulsion integration, and high-lift devices were carried out. A wide range of numerical methods ranging from lifting line, over unsteady vortex-lattice and surface vorticity panel methods to Reynolds-averaged Navier-Stokes computations were thereby utilized and compared with each other. The studies led to important design parameter sensitivities and recommendations that were subsequently fed back to the overall aircraft design. Moreover, the design studies yielded an increase of 10% in aerodynamic performance (L/D) and a reduction of -6% in the required propulsive power. Considering the slipstream effect during the high-lift design indicated a potential for improved climb performance.

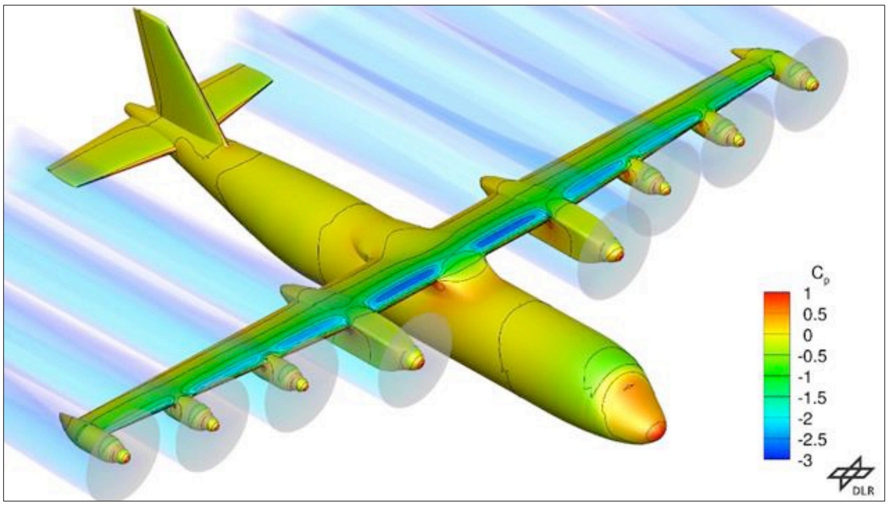


Figure 1: *Flow visualization of IMOTHEP regional propeller-driven transport aircraft concept with distributed propulsion computed with RANS*

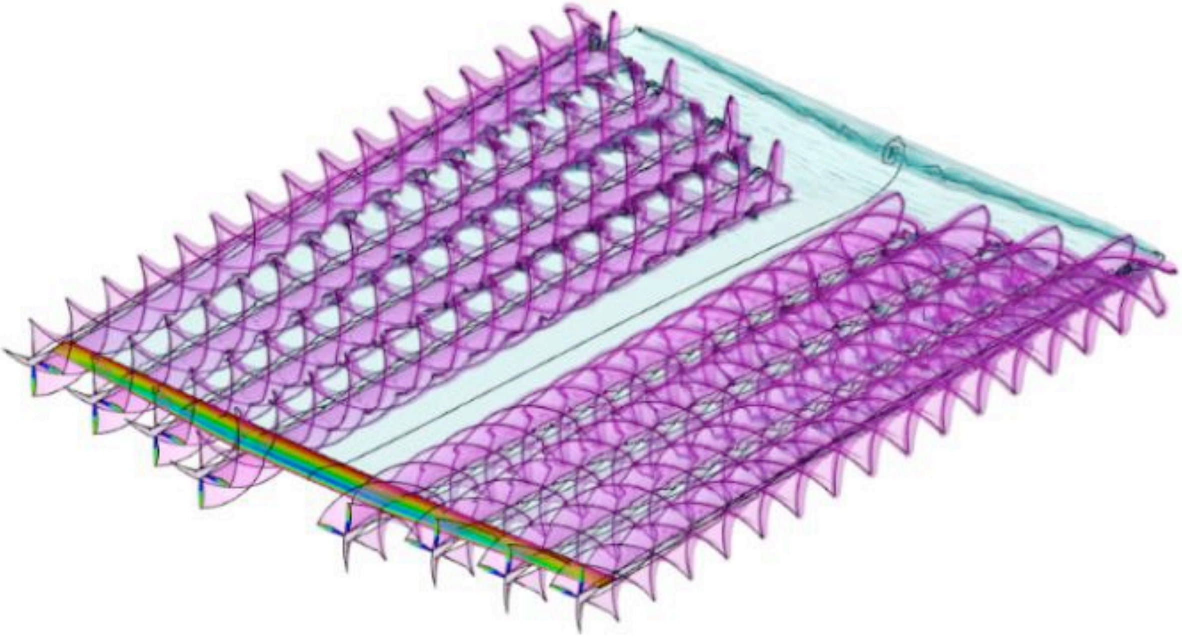


Figure 2: *Wake development from the wing + propellers configuration computed with unsteady VLM*

References

[1] G. Atanasov, „Plug-In Hybrid-Electric Regional Aircraft Concept for IMOTHEP“ in *12th EASN International Conference on Innovation in Aviation and Space for opening NewHorizons*, Barcelona, Spain, 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 240-5

Increasing Take-Off Performance of a Distributed Propulsion Wing Section Using Segmented Flaps

Till K. Lindner[1]*, Jonas Oldeweme[2], Peter Scholz[1] and Jens Friedrichs[2]

[1] Institute for Fluid Mechanics, Technical University Braunschweig, Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany, t.lindner@tu-braunschweig.de

[2] Institute of Jet Propulsion and Turbomachinery, Technical University Braunschweig, Hermann-Blenk-Str. 37, 38108 Braunschweig, Germany, j.oldeweme@ifas.tu-braunschweig.de

Key Words: *Distributed Propulsion, propeller-wing interaction, RANS, induced drag*

Electrically powered transport aircraft promise to be one answer to the need of energy efficient configurations. Independent of the source of electricity (turbo-electric/fuel cell/battery hybrid), at industry relevant transport capacity the integration of high power electric motors is both an opportunity and challenge for the air frame designer. Distributing multiple drive units along the wing reduces the power per propulsor to moderate levels. Such distributed propulsion (DP) concepts benefit from increased total disc area on the propulsion side and higher maximum lift of the wing. The benefits are opposed by the propeller-wing interaction leading to distorted lift distributions and additional drag from nacelles and slipstream velocity. Taking these parameters into account, a system efficiency of a periodic wing section with distributed propulsion is evaluated with RANS in this paper. The findings are backed up with wind tunnel data, verifying the RANS + actuator disc approach at high lift. On system level, the additional wing drag due to DP dominates the DP efficiency, while the propulsive efficiency alone is only slightly increased. Therefore, in this paper we strive to reduce the drag of the DP wing section at a constant thrust and lift requirement. While the viscous drag in the slipstream velocity can only be slightly reduced by a changing the radial blade loading of the propeller, the induced drag due to spanwise lift variation in the propeller wake is partially compensated for by deferentially changing the flap angle of the wing. The potential of such a spanwise varying DP wing section is evaluated in the final paper.

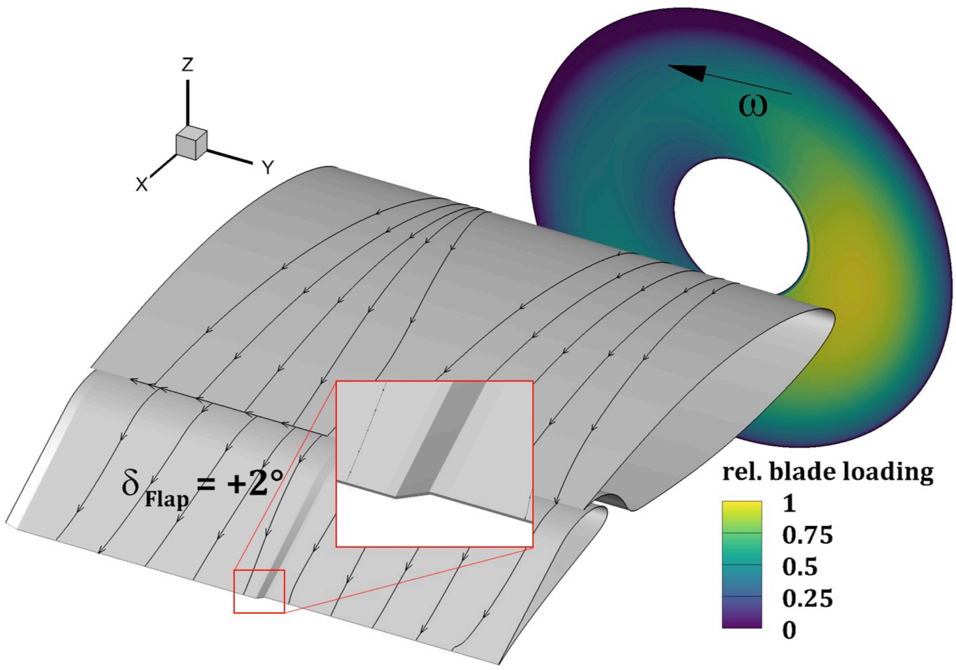


Figure 1: *Differential flap deflection to reduce spanwise circulation variation.*

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 240-6

Investigating Truss-Braced Wing Configuration through CFD Based Analysis

Mihai-Vlăduț Hothazie[1,2]*, Daniel-Eugeniu Crunțeanu[2], Ionuț Bunescu[1,2], Mihai-Victor Pricop[1,2]

[1] National Institute for Aerospace Research "Elie Carafoli" (INCAS), B-dul Iuliu Maniu, no. 220, sect. 6, 061126 Bucharest, Romania, https://www.incas.ro/, hothazie.mihai@incas.ro

[2] University Politehnica of Bucharest: Faculty of Aerospace Engineering, Street Polizu, no. 1, sect. 1, 011061 Bucharest, Romania, http://www.aero.pub.ro/en/home/, daniel.crunteanu@upb.ro

Key Words: *Strut-braced wing, high-fidelity, transonic, shock-wave, open-source, hyperbolic meshing, overset*

Meeting the demand for high-performance, aerodynamically efficient aircraft requires not only optimizing existing designs but also exploring novel configurations. Among these, the strut-braced wing configuration shows promise due to its potential for superior aerodynamic performance driven by an increased aspect ratio wing coupled with structural efficiency. However, designing and optimizing such a configuration requires the use of high-fidelity solvers, especially in regions where shock-wave interactions within the transonic regime occur. The computational demands, including time and power requirements, pose challenges in optimizing strut-braced wing configurations, necessitating the development of an efficient framework for aerodynamic performance evaluation. This paper aims to overcome these challenges by developing a framework based on high-fidelity open-source codes such as ADflow[1] and pyHyp[2]. Key to this framework is the integration of an effective hyperbolic meshing technique based on the overset method, in order to significantly reduce computational requirements and enable in-depth parametric studies and optimization processes tailored to the geometric characteristics of the wing and strut shape and positioning.

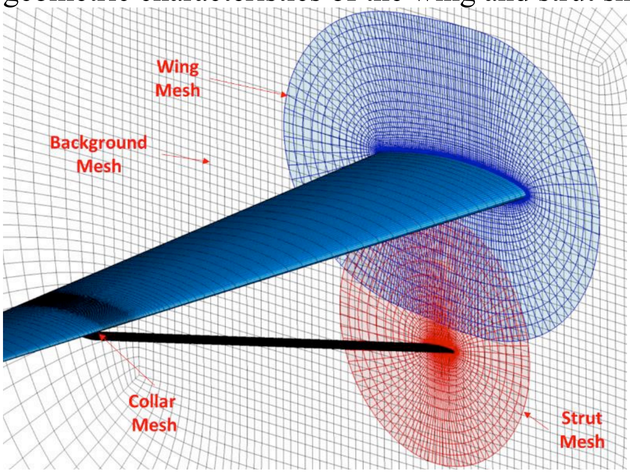


Figure 1: *Overset mesh technique*

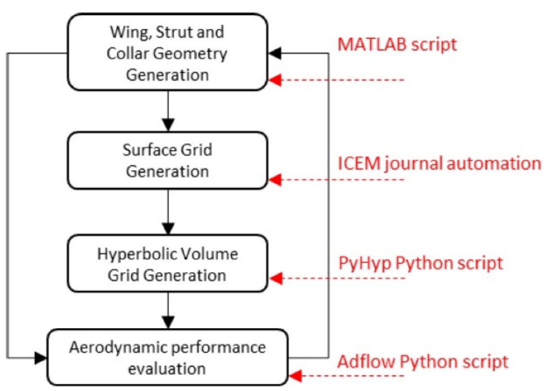


Figure 2: *Aerodynamic framework*

In this case, a cartesian mesh is initially created to serve as the background mesh. After that, hyperbolic meshes are then generated for the wing and strut, starting from an already generated structured multiblock surface mesh as can be seen in Figure 1. The grid generation approach employs hyperbolic grid generation techniques that utilize partial differential equations to create smooth, high-quality cells. This process ensures proper alignment of grid lines with the geometry. Additionally, a collar mesh near the wing-strut intersection enhances mesh quality and ensures valid overset operation. This automation procedure of the entire framework encompasses not only volume mesh generation but also geometry and surface grid generation, utilizing MATLAB and ICEM as can be seen in Figure2.

For analysing the aerodynamic performance, ADflow solves the Reynolds-averaged Navier-Stokes (RANS) equations coupled with the Spalart-Allmaras turbulence model. Steady-state flow simulations use a multi-grid approach, initializing with the approximate Newton-Krylov (ANK) algorithm for initial iterations and transitioning to the Newton-Krylov (NK) algorithm for final stages. Thus, this framework enables analyses that hold the potential to contribute with new aerodynamic insights beneficial for designing and optimizing the strut-braced wing configuration.

References

[1] C. A. Mader, G. K. W. Kenway, A. Yildirim, and J. R. R. A. Martins, "ADflow: An Open-Source Computational Fluid Dynamics Solver for Aerodynamic and Multidisciplinary Optimization," Journal of Aerospace Information Systems, vol. 17, no. 9, pp. 508–527, Sep. 2020, doi: 10.2514/1.I010796.

[2] N. R. Secco, G. K. W. Kenway, P. He, C. Mader, and J. R. R. A. Martins, "Efficient Mesh Generation and Deformation for Aerodynamic Shape Optimization," AIAA Journal, vol. 59, no. 4, pp. 1151–1168, Apr. 2021, doi: 10.2514/1.J059491.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 240-7

Fluidic Actuation for Gust Load Reduction on an Aircraft Wing

André Bauknecht[1,2]*, Florian Siebert[1,2] and Khalid Khalil[1,2]

1 TU Braunschweig, Institute of Fluid Mechanics, Braunschweig, Germany, a.bauknecht@tu-bs.de

2 Cluster of Excellence SE2A (Sustainable and Energy-Efficient Aviation), Braunschweig, Germany

Key Words: *Gust load alleviation, active flow control, computational fluid dynamics*

Active gust load alleviation can help realize reductions in structural weight and climate-relevant emissions of future transport aircraft by limiting peak aerodynamic loads experienced in flight. Fluidic concepts (Surface and Coandă jets) can help achieve this target as they have high lift control authority and are potentially faster than conventional control surfaces [1].

In this study we investigate the unsteady performance and gust load reduction capabilities of fluidic actuation concepts using unsteady Reynolds-averaged Navier–Stokes (URANS) simulations with the DLR TAU code. The fluidic concepts are implemented on a supercritical airfoil and their steady and unsteady lift change performance characterized and compared with that of a trailing edge flap. The results show that the fluidic actuators can deliver comparable gust load alleviation performance as the trailing edge flap for well-predicted gust encounters and outperform the flap for delayed gust detection.

Promising actuator configurations are implemented on swept wing sections and on the half-model of a mid-range aircraft of the Cluster of Excellence SE²A [2], to investigate the sensitivity of actuator performance to sweep angle and wing integration, as shown in Fig. 1. For a surface jet, for example, spanwise distributions of round air jets are compared with continuous and spanwise partitioned slots to identify the most suitable approach. Unsteady interactions with vertical gusts are simulated for the half-model based on the disturbance velocity approach and fluidic actuation is applied to counteract the gust-induced lift increase, as shown for the spanwise lift distribution in Fig. 2. The progress towards fluidic gust load alleviation on an elastic wing is presented, introducing results of URANS simulations coupled to a structural modal model of the wing box using the coupling environment IFLS [3].

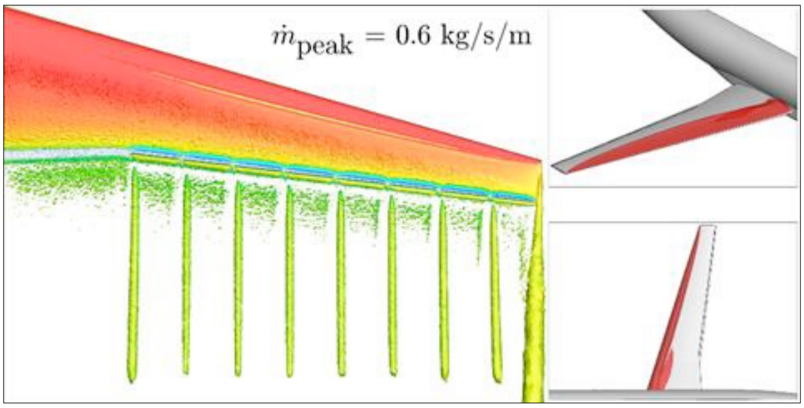


Fig. 1: *Extent of supersonic flow and blowing-induced vortices on actuated wing*

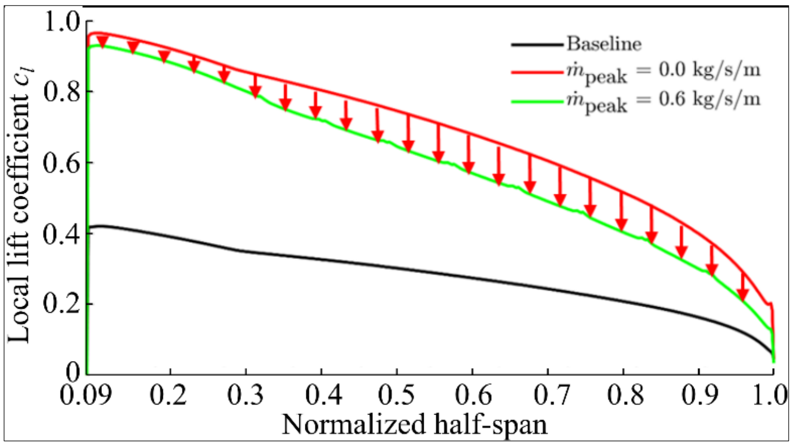


Fig. 2: *Spanwise lift distribution for gust encounter (red curve) and with gust load alleviation (green)*

Acknowledgements

We acknowledge the funding by Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) under Germany's Excellence Strategy EXC 2163/1 Sustainable and Energy Efficient Aviation Project ID 390881007, and the computing time granted by the Resource Allocation Board and provided on the supercomputer Lise and Emmy at NHR@ZIB and NHR@Göttingen as part of the NHR infrastructure. The calculations for this research were conducted with computing resources under the projects nii00166 and nii00178.

References

[1] K. Khalil, S. Asaro and A. Bauknecht, Active Flow Control Devices for Wing Load Alleviation, Journal of Aircraft, 59(2): 1–17, 2021. DOI: 10.2514/1.C036426

[2] S. Karpuk, V. Mosca, C. Liu and A. Elham, Development of a Multi-fidelity Design, Analysis, and Optimization Environment for Future Transport Aircraft, AIAA Scitech 2022 Forum, AIAA 2022-0686, San Diego, CA, USA, 2022. DOI: 10.2514/6.2022-0686.

[3] M. Haupt, R. Niesner, R. Unger and P. Horst, Coupling Techniques for Thermal and Mechanical Fluid-Structure-Interactions in Aeronautics. Proc. Appl. Math. Mech., 5: 19-22, 2005. DOI: 10.1002/pamm.200510006

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 240-8

Assessment of Engine/Airframe Aerodynamic Performance: Comparison between ONERA, DLR, NLR and Airbus

Fulvio Sartor[1]*, Arne Stuermer[2], Martin Laban[3], Stevie Ray Janssen[3] and Bastien Caruelle[4]

[1] ONERA, 8 rue des Vertugadins, 92190 Meudon, France, *fulvio.sartor@onera.fr*

[2] DLR Institute of Aerodynamics and Flow Technology, Braunschweig, Germany

[3] Netherlands Aerospace Centre NLR, 1059 CM Amsterdam, the Netherlands

[4] Airbus Operation SAS, Toulouse, France

Key Words: *CFD, rotating fan, installation effect, engine performance, code comparison*

As a partner in the EU funded Clean Sky 2 project ADEC (Large Passenger Aircraft platform, dedicated to innovative motorized aircraft configuration - Grant Agreement No 945583), ONERA, DLR and NLR are contributing with Airbus to an investigation of the capability of current advanced CFD approaches to reproduce the flow around a complex configuration, representative of an airplane equipped with a UHBR engine (Ultra High Bypass Ratio), installed under the wing.

Numerical simulations have been performed to reproduce the flow around the aircraft, including the installed engine and the rotating fan, by modelling both the external aerodynamics of the plane and the internal aerodynamics in the secondary flow. On this type of configurations, the interactions between the engine and the airframe are reinforced because of the large diameter of the engine and its proximity to the wing.

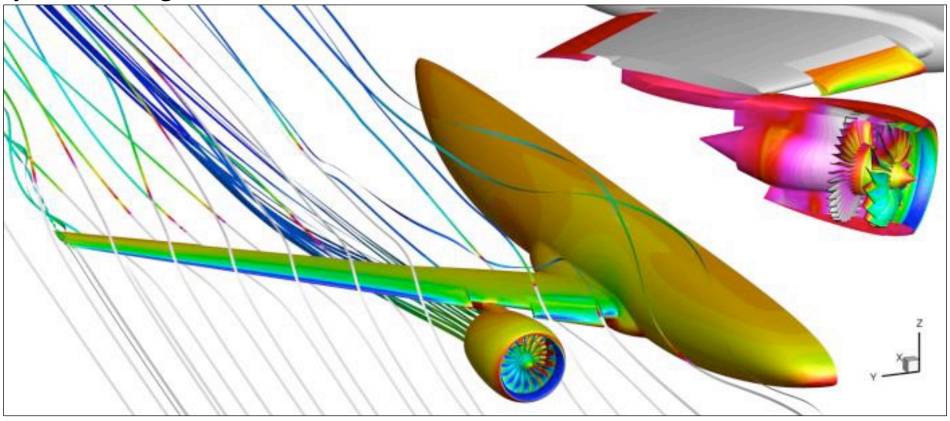


Figure 1: ***ONERA simulation in take-off conditions***

The identified test case, named NOVASPIRE, is the fusion between one of the NOVA configurations [1], the aircraft demonstrator designed by ONERA, with the ASPIRE engine [2], resulting from a previous collaboration between the partners [3].

The article will present a summary of the results of each partner, including a code-to-code comparison of 360° URANS computations needed to reproduce both the airframe, in cruise and high-lift conditions, and the nacelle with a geometrically fully modelled fan and OGV (Outlet Guide Vane). The code-to-code comparison is done between the structured solver of the *elsA* code, the *ENFLOW* solver (structured, developed by NLR) and the *TAU* solver (unstructured, developed by DLR).

References

[1] L. Wiart, O. Atinault, B. Paluch, D. Hue, R. Grenon. "Development of NOVA Aircraft Configurations for Large Engine Integration Studies", *AIAA 2015-2254*

[2] A. Burlot, F. Sartor, M. Vergez, M. Méheut, R. Barrier. "Method comparison for fan performance in short intake nacelle", *AIAA 2018-4204*

[3] M. Méheut, F. Sartor, M. Vergez, M. Laban, R. Schnell, A. Stuermer, G. Lefevre. "Assessment of Fan/Airframe aerodynamic performance using 360° uRANS computations: Code-to-Code comparison between ONERA, DLR, NLR and Airbus", *AIAA 2019-0582*

STS 243 A & B

Contributions of EU-Funded Projects towards Greener and Digital Transport

Chairs: Georgios Bampanis[1], Dietrich Knoerzer[2]

[1] Climate, Infrastructure and Environment Executive Agency - CINEA, European Commission, 1049 Brussels, Belgium, georgios.bampanis@ec.europa.eu

[2] Aeronautics Consultant, 1180 Brussels, Belgium, dietrich.knoerzer@outlook.com

Key Words: *Aviation, road, waterborne, automated transport, climate impact, emissions, energy efficiency, aerodynamics, numerical methods, digital twins, AI*

The "European Green Deal" [1] has set ambitious goals for the transport sector, calling for a 90% reduction in its greenhouse gas emissions by 2050. To achieve this systemic change, the European Commission has adopted the "Sustainable and Smart Mobility Strategy" [2] which aims to ensure that the EU transport sector is fit for a clean, digital and modern economy. Among others, this strategy indicates that all modes of transport should be made more sustainable, and outlines the need for the decarbonisation and energy efficiency improvement of aviation and maritime transport in particular. To this end, digitalisation and automation will become important drivers to deliver on these greening objectives and to maintain and reinforce the EU's leadership, and competitiveness. It is also essential that key digital enablers for design, manufacturing and automation are in place for all transport modes. This includes electronic components for mobility, network infrastructure, cloud-to-edge resources, data technologies and governance, digital twins, as well as Artificial Intelligence (AI).

Thanks to the continuously increasing capabilities of high-performance computing (HPC) hardware, digitalized design can facilitate the testing, certification and deployment of the innovative solutions required to minimize the environmental impact of airborne and waterborne transport. Further advancement of multi-disciplinary design optimization methodologies and simulation tools, along with integration of AI methods and big data analysis are thus important challenges to address, in order to reduce emissions during the industrial production process and the entire product lifecycle.

The European Commission's European Climate, Infrastructure and Environment Executive Agency (CINEA) is currently implementing a broad portfolio of collaborative R&I projects, funded under Horizon 2020 and Horizon Europe, which are contributing to the European Green Deal through the aforementioned specific technological challenges. These projects are developing and applying advanced methods for modelling, simulation, optimization and design of technologies contributing to the mitigation of the environmental impact in airborne, road and waterborne transport. The portfolio of EU-funded projects managed by CINEA is expected to be further enhanced by selecting new projects from upcoming calls for proposals within the Horizon Europe Cluster 5 on Climate, Energy and Mobility.

References

[1] European Commission, Communication from the Commission, The European Green Deal, COM(2019) 640 final, 2019.

[2] Sustainable and Smart Mobility Strategy – Putting European Transport on Track for the Future, COM/2020/789 final; https://eur-lex.europa.eu/legal-content/EN/TXT/?uri=CELEX%3A52020DC0789

Allocation of the papers to the session parts A and B:
Part A: STS 243-1 to STS 243-4; **Part B:** STS 243-5 to STS 243-8.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-1

The Dynamic Pickup and Delivery Problem with Crossdock for Perishable Goods

Emmanouil Nisyrios[1,2]*, Amalia Nikolopoulou[1,3], Konstantinos Gkiotsalitis[1,4]

[1] National Technical University of Athens, School of Civil Engineering, Dep. of Transportation Planning & Engineering, Iroon Polytechniou 5, 15773, Athens, Greece, www.ntua.gr/en/
[2] emnisyrios@mail.ntua.gr, [3] anikolopoulou@mail.ntua.gr, [4] kgkiotsalitis@civil.ntua.gr

Key Words: *Pickup and Delivery Problem, PDPCD for Perishable Goods, Fleet Management, Rolling-Horizon*

This research extends the original Pickup and Delivery Problem (PDP) framework [1] by introducing a Crossdock (CD) to facilitate the efficient exchange of goods exchange among connected vehicles, reducing the delivery routes and running times. Operational constraints including time windows, vehicle capacities, and crossdock synchronization are considered, along with specific requirements for perishable goods, aiming to reduce the distance of long trips. This extension, named the Pickup and Delivery Problem with Crossdock for Perishable Goods (PDPCDPG), is formulated as a nonlinear programming problem. Leveraging linearization and valid inequalities, a reformulation of the PDPCDPG is performed, resulting in a Mixed-Integer Linear Programming (MILP) model. This enables the computation of globally optimal solutions through Branch-and-Cut algorithms. Computational experiments on benchmark instances demonstrate the stability and effectiveness of the proposed approach.

Building upon this, in the context of the EU Horizon Europe project CONDUCTOR [2], we propose a time-based Rolling Horizon extension to enhance real-time route optimization, considering dynamic traffic conditions. Integration with the Aimsun traffic simulation software enables the application of both offline and online simulation approaches, incorporating real-world network dynamics. Through iterative updates of travel times and route scheduling, the PDPCDPG's performance is optimized within evolving traffic environments. This integration offers transportation and logistics companies a practical solution for time-sensitive deliveries, enhancing operational efficiency and responsiveness to real-world challenges.

Results demonstrate that vehicle routes are optimized in real-time, and travel times and overall costs are reduced. It is also expected to reduce the vehicle kilometres travelled and improve the quality of transportation company services currently offered. The project's innovations will therefore lead to smoother urban traffic, less pollution, and a higher quality of delivery services.

Acknowledgement

This work is partially funded by the European Union's Horizon Europe research and innovation programme CONDUCTOR (Grant Agreement No. 101077049).

References

[1] Savelsbergh, M. W., & Sol, M. (1995). The general pickup and delivery problem. Transportation science, 29(1), 17-29.

[2] CONDUCTOR Consortium (2022). CONDUCTOR – Advanced, high-level traffic and fleet management. Accessed: 13. 5. 2024. https://conductor-project.eu/.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-2

Traffic Forecasting with Uncertainty: A Case for Conformalized Quantile Regression

Gašper Petelin[1,3], Jože M. Rožanec[2,3,4] and Gregor Papa[1,3,*]

[1] Computer Systems Department, Jožef Stefan Institute, Ljubljana, Slovenia, gasper.petelin@ijs.si
[2] Laboratory of Artificial Intelligence, Jožef Institute, Ljubljana, Slovenia, joze.rozanec@ijs.si
[3] Jožef Stefan International Postgraduate School, Ljubljana, Slovenia, gregor.papa@ijs.si
[4] Qlector d.o.o., Ljubljana, Slovenia

Key Words: *Traffic forecasting, uncertainty estimation, and calibrated prediction intervals*

Accurate and reliable traffic flow forecasting is of primary importance for traffic planning and management. While there is a growing interest in real-time traffic forecasting models, accurate predictions remain a challenge due to the dynamic nature of traffic systems and the multiple factors that affect the traffic flow. Point forecasts do not provide insights regarding uncertainties associated with forecasts. Furthermore, many traffic flow models fail to produce prediction intervals that accurately capture the uncertainty of the forecasts. Therefore, we investigate the use of quantile regression models [1] for traffic flow forecasting and highlight their tendency to generate prediction intervals that are too narrow and poorly calibrated [2]. To address this issue, we propose using conformal predictions, which allow us to achieve well-calibrated prediction intervals leading to more accurate, reliable, and therefore trustworthy predictions.

Additionally, we show that using quantized conformal regression to calibrate machine learning models offers much more accurate predictions that do not deviate significantly even for longer forecasting horizons. This advanced approach enhances the robustness of the forecasts by effectively addressing the uncertainties inherent in the traffic system, thus providing a more reliable tool for traffic management and planning.

Acknowledgement

The authors acknowledge the financial support of the Slovenian Research and Innovation Agency under research core funding No. P2-0098. This work is also part of a project that has received funding from the European Union's Horizon Europe research and innovation programe under Grant Agreement No. 101077049 (CONDUCTOR).

References

[1] Yaniv Romano, Evan Patterson, and Emmanuel Candes. 2019. Conformalized quantile regression. Advances in neural information processing systems 32 (2019).

[2] Ying Wu, Yongchao Ye, Adnan Zeb, James Jianqiao Yu and Zheng Wang. 2023. Adaptive Modeling of Uncertainties for Traffic Forecasting. IEEE Transactions on Intelligent Transportation Systems (2023).

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-3

Fleet and Traffic Management Systems for Conducting Future Cooperative Mobility

Gregor Papa[1]*, Vida Vukašinović[2], Flavien Massi[3]

[1] Jožef Stefan Institute, Slovenia, [1] gregor.papa@ijs.si, [2] vida.vukasinovic@ijs.si
[3] Netcompany, Luxembourg, flavien.massi@netcompany.com

Key Words: *Transport engineering, traffic management, fleet management*

As ongoing urbanization increases worldwide, cities face the challenge of accommodating growing populations while maintaining efficient and sustainable transportation systems. The emergence of connected and autonomous vehicles promises transformative changes in urban mobility. This paper outlines developments and innovations that aim to seamlessly integrate autonomous vehicles into the complex ecosystem of urban mobility. This is part of the Horizon Europe project CONDUCTOR [1], which aims to design, integrate and demonstrate advanced traffic and fleet management that enables efficient and globally optimal transportation of passengers and goods, while ensuring seamless multimodality and interoperability, through dynamic balancing and priority-based management of automated and conventional vehicles. We build on the fleet and traffic management solutions in the Collaborative, Connected, and Automated Mobility (CCAM) ecosystem.

We will develop the next generation of multi-level simulation models and tools enabled by AI and data fusion. These will enhance the capabilities of transportation authorities and operators to become true conductors of future mobility networks. We will apply and test a portfolio of enhanced solutions in complementary environments in different regions. These include use cases in five European countries (Greece, Italy, Slovenia, Spain and the Netherlands):

Integrated Traffic Management (demonstrated in Athens, Madrid and Almelo), Demand-Responsive Transportation (between Slovenian cities and Italian airports) and Urban Logistics (in Madrid).

The CONDUCTOR project final results are expected to demonstrate that delays at traffic lights are reduced, door-to-door travel times are shortened and passengers have a better experience of public transport. It is also expected to improve the quality of transport services currently offered and reduce the length of traffic queues. The project's innovations will therefore lead to smoother urban transportation, less pollution and a higher quality of life.

The authors acknowledge the financial support of the Slovenian Research and Innovation Agency under research core funding No. P2-0098. This work is also part of a project that has received funding from the European Union’s Horizon Europe Research and Innovation Programme under Grant Agreement No. 101077049 (CONDUCTOR).

References

[1] CONDUCTOR Consortium (2022). CONDUCTOR – Advanced, high-level traffic and fleet management. Accessed: 10. 1. 2024. https://conductor-project.eu/.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 243-4

AI4CCAM: Trustworthy AI for Connected, Cooperative Automated Driving

J. Altman[12], S. Antoniou[2], C. Arlotti[7], N. Belmecheri[1], B. Brugger[6], L. Cantat[7], G. Ciccarelli[5], A. Cortés[2], L. B. Eklund[1], M. Eynaud[9], L. Franchi[5], J. Garcia Alfaro[4], Arnaud Gotlieb (Coordinator)[1,*], G. Gowrishankar[14], E. Güzel Kalayci[13], M. Jeudy[6], D. Kuzmina[8], N. Lazaar[14], E. Maudet[11], R. Meyfarth[10], M. Mischinger-Rodziewicz[13], N. Papakosta[2], R. Passama[14], K. Quintero[7], M. Schmeja[13], H. Spieker[1], A. Taillandier[4], M. Tlig[7], S. Trejo[2], L. Urcelay[2], R. Václavík[12], C. Van Hek[8], P. Vasishta[6] and S. Viktor[14]

[1] Simula Research Laboratory, Oslo, Norway, www.simula.no
[2] Inlecom, Kifissia Athens, Greece, www.inlecom.eu
[3] Barcelona Supercomputing Center, Barcelona, Spain, www.bsc.es
[4] SAMOVAR, Télécom Sud Paris, France, www.imt.fr
[5] TTS Italia, Roma, Italy, www.ttsitalia.it
[6] Akkodis, Guyancourt, France, www.akkodis.com
[7] Institute for Technological Research SystemX, Saclay, France, www.irt-systemx.fr
[8] The International Association of Public Transport, Brussels, Belgium, www.uitp.org
[9] BVA Xsight, Balma, France, www.bva-group.com
[10] Deep Safety, Berlin, Germany, www.deepsafety.ai
[11] Institut Mines-Telecom, Palaiseau, France, www.imt.fr
[12] Skoda Auto a.s., Mladá Boleslav, Czech Republic, www.skoda-auto.cz
[13] Virtual Vehicle Research GmbH, Graz, Austria, www.v2c2.at
[14] Centre National de la Recherche Scientifique, LIRMM, Montpellier, France, www.cnrs.fr
Contact: arnaud@simula.no

Key Words: *Trustworthy Artificial Intelligence; Automated Driving*

Artificial Intelligence is considered one of the key enabling technologies that can lead to the successful deployment of automated vehicles. By leveraging the huge amount of data collected by new and powerful sensors, artificial intelligence can revolutionise the future of automated driving functions. However, the benefits of artificial intelligence in automated driving are hampered by ethical risks that can compromise its adoption by car drivers, passengers or vulnerable road users such as pedestrians, cyclists, or persons with disabilities.

The EU-funded AI4CCAM project[1] aims to address this hurdle by developing trustworthy by-design artificial intelligence methods and models for automated driving functions. Leveraging the Trustworthy AI guidelines – a document created by the European Commission's High-Level Expert Group on Artificial Intelligence – and the ethical recommendations for automated driving scenarios with ethical risks involving vulnerable road users, (ii) the design of a trustworthy deep learning architecture which embeds pedestrian and cyclist behaviour anticipation models, (iii) scene understanding through qualitative constraint acquisition, (iv) visual gaze estimation and virtual reality, (v) user acceptance studies including levers and barriers for automated vehicles, (vi) simulate explainable car trajectory prediction models.

Taking into account all the capabilities and potential risks of AI, AI4CCAM creates trustworthy AI models and methods that will advance the safety and user acceptance of automated vehicles. These models will be tested on scenarios illustrated in three distinct use cases which will cover the overall chain Sense-Plan-Act of automated vehicles, as well as the necessary user acceptance questions that naturally arise when AI models and methods are used. The principles of trustworthiness identified at a European level will be enriched and transposed into specific indicators and practices enabling the evaluation of the systems under test.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-5

OPTIWISE: Design Optimisation of Wind Assisted Ship Propulsion

Maarten Flikkema[1]*, Tom van Terwisga[2], Rogier Eggers[2], Maxime Garenaux[2], Sofia Werner[3] and Gabriele Mazza[3]

[1] Flikkema Innovation Management and Consultancy, Bantpolder 56, 3825JN, Amersfoort, The Netherlands, MF@Flikkema.eu, www.flikkema.eu

[2] MARIN, Haagsteeg 2, 6709PM Wageningen, The Netherlands, T.v.Terwisga@marin.nl, www.marin.nl

[3] RISE, Chalmers tvärgata 10, 412 58 Göteborg, Sweden, Sofia.Werner@ri.se, www.ri.se

Key Words: *CFD, RANS, Hydrodynamics, Ship design, Wind assisted propulsion*

The maritime shipping sector is responsible for nearly 3% of the global GHG emissions [1] and have set ambitious goals to reduce the emissions to net zero emissions around 2050 [2]. This can only be achieved by a combination of clean fuels and energy reduction. Energy density characteristics of clean fuels cause the need for more energy storage space on board or take up more weight.

OPTIWISE set out to accelerate the market implementation of Wind assisted ship propulsion (WASP) to achieve fuel reduction, by: 1. improving ship design methods for ships with WASP; 2. demonstrating three applications of three WASP solutions (rotor sails, Ocean wings, and solid sails); 3. promoting the energy saving potential of WASP. Maritime research institutes MARIN and RISE employ state-of-the-art numerical models to address the complex interactions involved in wind-assisted propulsion systems. Numerical simulations play a pivotal role in optimizing the design and performance of these innovative technologies, offering a detailed understanding of their aerodynamic, hydrodynamic, and structural dynamics.

MARIN's holistic optimization tool CREATOR encompasses global design aspects of WASP vessel, including hull design, cargo carrying capacity, powering in calm water, manoeuvring, seakeeping, and aerodynamic performance. The basis of the global ship design lies in a parametric ship model, supported by surrogate models generated through machine learning. These models leverage high-fidelity tools such as RANS, potential flow, lifting line and model testing. From the performance prediction, the required power profile is derived and used for the engine room optimisation integrated into the overall design process. An optimisation loop is performed around CREATOR to find the optimum in terms of for example transport efficiency, while satisfying various constraints such as limiting ship motions or minimum cargo capacity.

RISE employs the in-house developed software SEAMAN Wind investment decision support as base for the design process. The ship performance and the voyage & routing modules of SEAMAN Winds allows predicting in an effective way the KPIs of a WASP vessel. The ship performance uses hydrodynamic manoeuvring derivatives determined using RANS simulations coupled with efficient aerodynamic models. The 'polars', output from the ship performance module, are applied in the voyage module to the intended trading route and weather statistics.

References

[1] "Fourth IMO GHG Study 2020," no. MEPC 75/7/15. International Maritime Organization (IMO); Marine Environment Protection Committee, London, United Kingdom, 2020

[2] "2023 Imo Strategy on Reduction of GHG Emissions from Ships", No. MEPC 80/17/Add.1 Annex 15, Marine Environment Protection Committee, London, United Kingdom, 2023

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-6

Numerical Simulation of Microwires within Carbon-Fibre Composites for Structural Health Monitoring of Aerospace Components

Wolfgang Krach[1,*], Walter Vonach[1], Johan Malm[2], Christer Johansson[2] and Arkadi Zhukov[3]

[1] CAE Simulation & Solutions, Pitkagasse 2/1/16, 1210 Vienna, krach@cae-sim-sol.at

[2] RISE Research Institutes of Sweden, Digital Systems, 41133 Göteborg, Sweden, johan.malm@ri.se

[3] Department Advanced Polymers and Materials and Dept Applied Physics, Univ. Basque Country, UPV/EHU, San Sebastian, Spain, arkadi.joukov@ehu.es

Key Words: *Electromagnetic simulations, mechanical simulations, finite element method, microwires, lifetime prediction, wireless measuring, CFRP composite, SHM*

The usage of carbon-fibre reinforced composites (CFRP) to achieve lightweight structures has further spread throughout the last years in aerospace industry. The lifetime prediction and structural health monitoring (SHM) of these structures is necessary and costly. The usage of magnetic microwires embedded within the CFRP is aimed to enable wireless SHM of the components in service.

In our setup, cobalt-rich magnetic microwires covered by a glass coating are used within noncrimp fibre (NCF) composites. These magnetic microwires change their electromagnetic properties while being thermally or mechanically loaded. This change can be detected and measured wirelessly using for instance a handheld high frequency (HF) reader system. To achieve a robust signal, electromagnetic simulations are carried out in parallel to experimental work to understand the physical basics of the interaction between CFRP and the microwires regarding electromagnetic fields. Unidirectional CFRP shows high orthotropy regarding electromagnetic properties. Industrially far more relevant multi-ply multi-directional CFRP shows quasi-isotropic electromagnetic behaviour, which is favourable to the accuracy of the measurements. A comparison of attenuation at 2.45GHz shows good correlation between simulations and measurements.

To check for practical industrial usability, mechanical simulations (FEM) are performed to ensure that the measuring range covers the generally used allowable strains of up to 0.4%. While accounting for the thermal mismatch in the production process, the orthotropic mechanical material properties of multi-ply CFRP and microwires as well as the rupture strains of the microwires, the simulations show that measuring ranges of up to 1.5% are to be expected.

The results indicate that microwires can be used for SHM. Additional work will be carried out to achieve more robust signals and to allow a quantitative signal assessment regarding different failure modes of CFRP.

The work is carried out within the Horizon Europe framework (HORIZON-CL5-2021-D5-01) “INFINITE - Aerospace Composites digitally sensorised from manufacturing to end-of-life”

References

[1] M. Ipatov, G.R. Aranda, V. Zhukova, L.V. Panina, J. González and A. Zhukov, “Tunable effective permittivity of composites based on ferromagnetic microwires with high magneto-impedance effect”, Appl. Phys. A: Mater. Sci. Process., Volume: 103(3): 693-697, 2011

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 243-7

On the Training of Algorithms Using Finite-Element Computation Data for Damage Identification in Sensorised Composite Structures

Rohan Chabukswar[†1]*, Chloe Mullen[†2] and Kostas Kouramas[†3]

[†] Collins Aerospace, Penrose Wharf Business Centre, Cork T23 XN53, Ireland
[1] rohan.chabukswar@collins.com, https://chabukswar.ie/
[2] chloe.mullen@collins.com, https://www.linkedin.com/in/chloe-mullen-ai/
[3] konstantinos.kouramas@collins.com, https://www.linkedin.com/in/kostaskouramas/

Key Words: *Structural health monitoring, embedded microwires, machine learning, intelligent structure.*

Embedding microwires into composite materials is a novel technology that can provide in-situ and remote data on the structural health of the system [1]. Damages such as delamination, disbond and cracks in the composite structure induce a change in the stresses and strains on the microwires, modifying their electromagnetic response [2]. Analysing this response with artificial intelligence can potentially allow us to not only detect these damages, but also characterise, localise, and quantify the damage, and possibly predict the remaining useful life of the structure.

As part of the Horizon Europe project INFINITE, this work aims to develop methodologies for in-service structural health monitoring of composites equipped with microwire sensors. The suitable machine learning algorithms include physics-aware three-dimensional convolutional neural networks, decision trees, logistical regression, etc., all of which require a large amount of training data, as accurate estimation on training including data instances of all type and levels. Since experimental data is scarce due to the low readiness-level of the technology, these algorithms are trained using finite-element numerical computation, which itself has a high computational cost, while fine-tuning is left to experimental data. Currently, maintenance, repair, and overhaul activities represent around 10–15% of an airline's operational costs [3], which in 2021 meant a global average of USD 234 million per airline [4]. In-situ monitoring of structural health of composite aircraft components will enable the aerospace industry to move from schedule-based maintenance to condition-based maintenance, especially for inaccessible components, by simplifying diagnostics and efficient logistics of repair operations.

This work was funded by the European Union under the Horizon Europe grant 101056884.

References

[1] Zhukov, Arcady P., Mihail Ipatov, Paula Corte-Leon, Juan M. Blanco, and Valentina Zhukova. "Advanced Magnetic Microwires for Sensing Applications," Encyclopedia of Materials: Electronics, (2023): 694-721.

[2] Allue, Alexandra, Paula Corte-León, Koldo Gondra, Valentina Zhukova, Mihail Ipatov, Juan Maria Blanco, Julian Gonzalez, Margarita Churyukanova, Sergey Taskaev and Arcady Zhukov. "Smart composites with embedded magnetic microwire inclusions allowing non-contact stresses and temperature monitoring", Composites Part A: Applied Science and Manufacturing 120 (2019): 12-20.

[3] J.P. Sprong & X. Jiang & H. Polinder. "Deployment of Prognostics to Optimize Aircraft Maintenance – A Literature Review", Journal of International Business Research and Marketing, Inovatus Services Ltd., (2020): 26-37.

[4] International Air Transport Association (IATA), "Airline Maintenance Cost Executive Commentary" (2022).

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3-7 June 2024, Lisbon, Portugal

STS 243-8

Deep Reinforcement Learning Applied to a Navigation Task in a Representative Flow Field for Urban Environments

Federica Tonti[1,*], Jean Rabault[2] and Ricardo Vinuesa[3]

[1] KTH Royal Institute of Technology, Engineering Mechanics, Osquars backe 18,
11428 Stockholm, Sweden, [1] ftonti@kth.se, [3] rvinuesa@mech.kth.se
[2] Independent Researcher, 0854 Oslo, Norway, jean.rblt@gmail.com

Key Words: *Deep Reinforcement Learning, Trajectory Optimization, UAVs, Navigation, Urban Flow*

The increasing number of Unmanned Aerial Vehicles (UAV) in urban environments requires a strategy to minimize their environmental impact, both in terms of carbon foot- print and noise reduction. In order to minimize these concerns, use of high-performance accelerated AI methodologies that empower innovative prediction models and optimization of flight planning with deep reinforcement learning (DRL) is needed.

Our goal is to develop DRL algorithms which are able to make the UAVs navigate autonomously in urban environments, taking into account the presence of buildings and other UAVs, optimizing the trajectories in order to reduce both emissions and noise. This will be achieved using CFD simulations which represent the environment in which UAVs navigate and training the UAV as an agent interacting with an urban environment and its complexities.

In this work, we show the application of DRL to planes extracted from CFD simulations of an urban environment represented by a two-dimensional flow field with the presence of two-dimensional obstacles, representing ideally buildings which are present in the path of the UAV. The presented methodology was validated by reproducing a simple but funda- mental problem in navigation, namely the Zermelo's problem [1] which deals with a vessel navigating in a turbulent ow, travelling from a starting to a target point optimizing the trajectory. This is the first step towards DRL strategies which will guide UAVs in a three-dimensional flow field using real-time signals, making the navigation efficient not only in terms of flight time and avoiding damages to the vehicle, but also reducing the noise produced which would increase the level of acoustic pollution in cities.

References

[1] Biferale, L. and Bonaccorso, F. and Buzzicotti, M. and Clark Di Leoni, P. and Gustavsson, K., Zermelos problem: Optimal point-to-point navigation in 2D turbulent flows using reinforcement learning, Chaos: An Interdisciplinary Journal of Nonlinear Science, AIP Publishing, 2019

STS 244

The Leading-Edge Computational Methods for Green Aviation

Chair: Wen BAI

Chinese Aeronautical Establishment, Aviation S&T Building, Beiyuan Road No.58, Chaoyang District, Beijing 10012, People Republic of China, https://www.cae.ac.cn/
baiwen@cae.ac.cn

Session Abstract

Key Words: *Computational Aerodynamics, Computational Structure, Artificial Intelligence (AI), Model-Based Systems Engineering, New Energy Air Vehicles.*

The STS, organized by Chinese Aeronautical Establishment (CAE), will address the most recently developed computational methods and its applications in aeronautical science and technology.
The topics include:

– Computational aerodynamics,

– Computational structure and material mechanics,

– Applications of Artificial Intelligence (AI) in numerical simulations,

– Model-Based Systems Engineering (MBSE) coupled MDAO, and the

– Concepts of new energy air vehicle,

Special aspects will be the applications for the global Climate-Neutral Aviation.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 244-1

Research on Airborne System Simulation Methodology Based on AI Enhanced Surrogate Modelling Approach

Chuang Wang[1]*, Wen Zhao[1]

[1] AVICAS GENERIC TECHNOLOGY CO., LTD, Jiangsu, Yangzhou, 225000, China

Key Words: *Airborne System, Virtual Integration, Complex Model Computation, Surrogate model, Large Language Models*

Model-based system engineering is a promising approach to achieve the design goal of green aviation by running virtual simulation to reduce energy consumption and emission. The virtual simulation of airborne systems encompasses the logic simulation of avionics and electromechanical controllers, as well as functional simulations of subsystems such as power system and hydraulics system [1]. Certain complex electromagnetic and fluid equations involved in this simulation possess characteristics of high computational costs and small convergence solving steps. Such inconsistency in the timescales of the entire simulation system results in a delay in the overall numerical simulation time. A common solution to address this issue is to employ surrogate models for model reduction [2].

The current model reduction methods used in airborne system integrated simulations in practical engineering tasks remain at the interpolation table stage, with low data complexity. However, most airborne system or module models are multidimensional, making it challenging to construct high-dimensional surrogate models. To tackle these challenges, this paper proposes a semi-automatic model reduction method based on large language models and existing mature fitting algorithms. A complete toolchain for model reduction has been developed, firstly, utilizing large language models to assist in selecting fitting algorithms and constructing fitted models. Then, training and deployment of models are conducted using an artificial intelligence model development platform, which is integrated into the GvSimLab platform to connect to other airborne system models for joint simulation.

Taking a particular IPMSM module (Internal Permanent Magnet Synchronous Motor) in airborne VFG system (Variable Frequency Generator system) as an example, compared to the currently used interpolation tools, this toolchain improves the accuracy of simulating nonsample data by 15%. Compared with the Simulink model, this toolchain can elevate the modeling-loop solving time from 30s per thousand steps to nearly real-time at 0.8s with an accuracy of 96.4%.

References

[1] W. Shenglai, W. Chuang , L. Yang , et al. Design and Implementation of Digital Integration Platform for Airborne System. Command Control and Simulation, https://link.cnki.net/urlid/32.1759.TJ.20231220.1402.008. 2023

[2] N. Delaissé, P. Havaej, D. Fauconnier and J. Degroote. Accelerating the simulation of elastohydrodynamic lubrication in a line contact using a surrogate model. In 8th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS Congress 2022). 2022

STS 244-2

Research on System Virtual Integration Method for Complex System

Fuxing TAO[1,*], Xiaosu MEN[2]

AVIC Digital Corporation Ltd, Building 19D, Compound A5, Shuguangxili,
Chaoyang District, Beijing, China,
[1] E-mail: magic10615@163.com, [2] E-mail: 3158680690@qq.com

Key Words: *Complex System, virtual integration framework, system emergence*

The increase in system complexity has brought about a paradigm shift in research and development. In order to accurately predict and evaluate the performance, characteristics, behaviors, and functions of the system, it is necessary to use simulation methods to integrate the key systems that make up the system virtually and carry out simulation verification in combination with the scenario environment to ensure that the characteristics and properties of the system emergence correctly through the interaction of multiple systems, thereby exposing design problems and faults at the front end and ensuring that the design is correct once and for all. This study proposes a new system virtual integration framework that adopts systems engineering methodology and hierarchical abstraction methods. The first layer verifies the behavior, functions, and interfaces of the system architecture model through discrete event simulation. The second layer constructs a multidisciplinary coupled dynamic model based on physical principles to verify the performance, spatiotemporal relationships, and other aspects of the system. The aim is to achieve design synthesis, virtual testing, requirement verification, and scheme iteration optimization of the system in the early stages of design, thereby improving the digital level of equipment research and development.

References

[1] H. Feiler, Jorgen Hansson, Dionisio de Niz, Lutz Wrage, System Architecture Virtual Integration: An Industrial Case Study, Technical Report, CMU/SEI-2009-TR-017, ESC-TR- 2009-017, 2009.

[2] H. W. Wang, H. E. Zhang. A Distributed and Interactive System to Integrated Design and Simulation for Collaborative Product Development. Robotics and Computer-Integrated Manufacturing, December 2010, 26(6):778-789

[3] Xin Huaisheng, Gao Bin, Li Lichao. Simulation Integration Based on HLA and DDS[J]. Communication Technology, 2011, 07(44): 33-36

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 244-3

A Class of Scalloped Riblets for Turbulent Drag Reduction

Yiqian Wang[1,*] and Song Fu[2]

[1] School of Mathematical Sciences, Soochow University, Suzhou 215006, China,
yiqian@suda.edu.cn

[2] School of Aerospace Engineering, Tsinghua University, Beijing 100084, China,
fs-dem@tsinghua.edu.cn

Key Words: *Scalloped Riblets, drag reduction, tip sharpness*

Riblets are small protruding surfaces aligned the direction of the flow, and are one of the most well-known passive turbulent drag reduction methods. We consider a class scalloped riblet, the shape of which was constructed by smoothly connecting two third-order polynomials and was not as sharp in the tip as corresponding triangular riblets with the same height-width ratio. Numerical simulations were then performed for turbulent channel flow with and without riblet control at various configurations. The drag reduction rates are obtained and compared. It is found that the drag reduction rate of a selected scalloped riblet is larger than the corresponding drag reduction rates of corresponding triangular riblets. Mean flow fields and second-order statistics of velocity, vorticity, and Liutex, a quantity introduced to represent vortices, were reported. It was found that streamwise vortices just above the riblet tips, which have a length scale of $200-300$ in wall units, have an important influence on those statistics and thus the turbulence generation cycle and the drag reduction mechanism. Pre-multiplied energy spectra of streamwise velocity and the Liutex component were reported to reveal the length scales in the flow field. Instantaneous vortical flow fields visualized by the Liutex method were provided with emphases on the streamwise vortices just above riblet tips. It should be noted that the class of scalloped riblets is suitable for investigations on the influences of curvatures at the tip and the valley of the riblet in future.

References

[1] P. Luchini, F. Manzo, and A. Pozzi, Resistance of a grooved surface to parallel flow and cross-flow, Journal of Fluid Mechanics, 228: 87-109, 1991.

[2] Y. Wang, Y. Huang and S. Fu, On the tip sharpness of riblets for turbulent drag reduction, Acta Mechanica Sinica, 38: 321389, 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 244-4

Optimization Design of Aircraft Structure Based on Strain Neighborhood Genetic Algorithm

Rui Zhang[1]*, Degang Cui[2] and Kaijian Wang[3]

[1] Chinese Aeronautical Establishment, Aviation S&T Building, Beiyuan Road No.58, Chaoyang District, Beijing, China, 10012, zhangrui@cae.ac.cn, https://www.cae.ac.cn/

[2] S&T Committee of AVIC, Beijing, China, dgcui@vip.163.com

[3] Chinese Aeronautical Establishment, Beijing, China, wangkaijian@cae.ac.cn

Key Words: *Strain neighborhood, Genetic algorithm, Large-scale design variables, Wing structure optimization, Lightweight design*

When the stiffness of composite materials varies in space, the tailoring and optimization of structure performance can be achieved. For high aspect ratio wing, the aerodynamic load of the wings is gradually transmitted from the tip through the skin, rib, spar to the root. Due to the design constraints of wing loading and leading and trailing edge control surfaces, the layout parameters of the wing beam vary less. Based on the weight proportion of the wing spar, rib and skin, the skin area accounts for a larger proportion. Therefore, optimization of the skin stiffness can achieve significant improvement space and weight reduction. Large scale design variable optimization expands the design space and can achieve more refined optimization design. However, at the same time, the design scheme has poor processability due to the lack of constraints on the process during design. In particular, large-scale design variables often use the global optimization algorithm, and the optimization results are decentralized, unable to meet the requirements of manufacture, and cannot achieve the continuity of force transmission.

This article proposes a genetic algorithm based on strain neighborhood and applies it to the structural optimization of composite material wings, achieving a reduction in structural weight while reducing sudden changes in skin ply thickness. The strain neighborhood based method proposed in this article considers the strain of the upper and lower wing finite element elements as a dataset. In these datasets, several higher strains will appear, making these elements cluster centers. Based on the position of the element where these higher strains are located as the center, according to the principle of four connections, the element with monotonic changes in strain values and common edges will be divided into a "neighborhood cluster", multiple neighborhoods can be established based on the strain range. The genetic algorithm optimization is based on the "neighborhood cluster" as the object. The units within the cluster are optimized as a whole, reducing the number of design variables and thickness variations between adjacent units, and also reflecting the continuity of force transmission, improving the continuity and smoothness of the skin laminate.

References

[1] A. C. Gray and J. R. Martins, "Geometrically Nonlinear High-fidelity Aerostructural Optimization for Highly Flexible Wings," presented at the AIAA Scitech 2021 Forum, 2021.

[2] G. Kenway, G. Kennedy, and J. R. R. A. Martins, "Aerostructural optimization of the Common Research Model configuration," presented at the 15th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference, 2014.

[3] B. K. Stanford, C. V. Jutte, and C. A. Coker, "Aeroelastic Sizing and Layout Design of a Wingbox through Nested Optimization," AIAA Journal, vol. 57, no. 2, pp. 848-857, 2019

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 244-5

Conceptual Design of Electric Distributed Propulsion Commuter Airplane

Haitao ZHU[1]*, Lefeng LV[2], Yuhang WANG[3] and Yan LI[4]

Chinese Aeronautical Establishment, Aviation S&T Building, Beiyuan Road No.58, Chaoyang District, Beijing 10012, China, https://www.cae.ac.cn/
[1] zhuhaitao@cae.ac.cn, [2] lvlefeng@cae.ac.cn, [3] wangyuhang@cae.ac.cn, [4] liyan@cae.ac.cn

Key Words: *Distributed propulsion, electric commuter airplane, battery energy density, extreme range, high-lift propeller, airplane-propulsion integration design*

Green aviation is one of the most prosperous areas of civil aviation. Subjected to both battery energy density and motor power density, electric propulsion will be used in general aircraft at first stage of electric civil aviation. As to battery and motor technology progresses in the near future, conceptual design of distributed commuter airplane is performed aim to remote region transport in this paper.

Firstly, the range and payload of electric airplane are studied according to battery energy density and lift-drag ration. Results show that, with battery power density 400~500wh/kg, the range will be 300~500km. if battery power density is 800wh/kg, the range will be 600~1000km. However, without breakthrough in battery area, main range of electric airplane is limited to 500~1000km and extreme range is less than 1500km.

Secondly, integration design of airplane and propulsion is studied with respect to electric distributed propulsion configuration. The distributed propulsion system consists of many highlift propellers which pulls aircraft and enhances downstream wing lift. Results show that, induced velocity distribution along blade radial direction has significant effect on wing lift increment. Besides, motor power and flight velocity are critical to the proportion of lift increment to wing lift without high-lift propeller slipstream. Increasing motor power, the proportion increases too. While, increasing flight velocity, the proportion decreases. Both highlift propeller motor power and flight velocity should be decided by the aircraft overall design.

At last, a conceptual scheme of commuter airplane aimed to remote region transport is depicted with total take-off weight 1400kg, range 500km, payload 250kg. Battery energy density 400wh/kg is used and lift-drag ration of airplane is 16 in the conceptual scheme. Distributed propulsion system consists of 10 high-lift propellers with 10kw motor power and a 120kw main propeller. And the high-lift propellers are operated during take-off and landing phases.

References

[1] J. Huang and F. T. Yang. Development and challenges of new energy electric aircraft. Acta Aeronautica et Astronautica Sinica, 37(1):57-68, 2016(in Chinese).

[2] Airbus reveals new zero-emission concept aircraft，https://www.airbus.com/en/newsroom/press-releases/2020-09-airbus-reveals-new-zeroemission-concept-aircraft

STS 245A & B

Prospects for Green Aircraft: Critical Technologies and Operational Aspects Driving Overall Design

Chair: Luiz Manuel Braga da Costa Campos

Departamento de Engenharia Mecânica, Instituto Superior Técnico, 1049-001 Lisboa, Portugal
luis.campos@tecnico.ulisboa.pt

Session Abstract

Key Words: *Green aircraft, hybrid turboelectric propulsion, regional aircraft, hydrogen, critical technologies*

The STS is based on the project EFACA (Environmentally Friendly Aviation for All Classes of Aircraft) that is supported by the European union in the period 2023 – 2026. It is intended to give a global overview of the challenges associated with the greening of aviation, taking as example an 80-seat 1000-km regional airliner with hybrid turboelectric propulsion, and including (i) overall design, (ii) critical technologies, (iii) economic and market prospects and (iv) environmental benefits.

The STS consists of presentations addressing the following topics:

- Overview of all aspects of the project EFACA, with focus on those aspects that have been selected for the following seven presentations in this STS.
- Preliminary design of an 80-seat 1000 km range regional airliner with hybrid turboelectric propulsion (HTEP) consisting of gas turbine and hydrogen fuel cell.
- The most innovative element of the electrified regional airliner, namely the HTEP system as concerns components and integration.
- One of the critical technologies for HTEP, concerning cryogenic fuel tanks for liquid hydrogen bearing in mind their larger volume compared with traditional fossil fuels for the same range.
- Another of the critical technologies for HTEP, namely hydrogen fuel cells with particular challenges of cooling and efficiency at altitude for aeronautical applications.
- Market prospects of the regional aircraft using HTEP as a replacement for existing fossil fuelled airliners, bearing in mind the additional complexity (preceding presentations) and environmental benefits (following presentations) and their impact on economics and operations.
- Benefits of a regional airliner using HTEP as concerns reductions of environmental impact in terms of emissions and noise at airport and global level.
- Sustainable Aviation Fuels (SAFs) focusing on alternative technologies for the production of alternatives to current fossil fuels.

Contributing papers to the session parts A and B:
Part A: STS 245-1 to STS 245-5; **Part B:** STS 245-6 to STS 245-9;

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-1

The EFACA Project – Environmentally Friendly Aviation

Luiz B. Campos[1], Joaquim G. Marques[2] and Pedro A. Serrão[3,*]

[1] Universidade de Lisboa, Instituto Superior Técnico, Lisboa, Portugal, luis.campos@tecnico.ulisboa.pt

[2] Universidade de Évora, Escola de Ciências e Tecnologia, Évora, Portugal, jmgmarques@uevora.pt

[3] Universidade de Lisboa, Instituto Superior Técnico, Lisboa, Portugal, pedro.a.serrao@tecnico.ulisboa.pt

Key Words: *EFACA project, Green Aviation, Hybrid propulsion, Hydrogen fuel, SAF*

An overview of Project EFACA (Environmentally Friendly Aviation for all Classes of Aircraft) is given focusing on the Special Technical Session topics.

The EFACA project [1] supported by the European Union in the period 2023-2026, consists of six main objectives taking into consideration three TRL3 demonstrations relevant to the greening of aviation; a preliminary design of propeller and jet driven airliners using hydrogen fuel; and a road map for achieving the EU environmental objectives for aviation.

Within the three TRL3 demonstrations of high relevance to the greening of aviation and the development of preliminary aircraft design the project involves:

1. Design, development and test a possibly world-first novel gearbox combining power inputs from a Gas Turbine Engine (GTE) and an Electric Engine (EE);
2. Improve performance and operating range of Hydrogen Fuel Cells (HFC) for aviation through a new phase cooling method;
3. Demonstrate a complete liquid hydrogen fuel system from cryogenic tank to fuel transfer, vaporization and combustion;
4. Design a regional aircraft using hybrid turbo-electric propulsion (HTEP) combining a gas turbine (GTE) and electric (EE) engines powered by an Hydrogen Fuel Cell (HFC);
5. Design a jet liner with using Liquid Hydrogen (LH) Fuel including all the cryogenic systems;
6. Build a global road map to achieve the EU Environmental Objectives for Aviation relative to 2035 and 2050, taking into account:
   a) emerging propulsion technologies (batteries, fuel cells, turbines) and sustainable fuels (synthetic aviation fuels and liquid hydrogen).
   b) application to all classes of aircraft: urban air mobility/helicopters/convertibles, regional/business, medium, and long-range airliners.

References

[1] https://efaca.eu/

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-2

Design of Hybrid Regional Airliner

Sergii Fil[1], Dmytro Berbenets, Andrii Khaustov, Oleksandra Urban*, Oleksandr Bondarchuk[1]

[1] Antonov Company, Tupoleva str. 1, 03062 Kiev, Ukraine fil@antonov.com

Key Words: *Hybrid Turbo-Electrical Propulsion, HTEP, regional passenger aircraft, environmentally friendly aviation*

One of the promising ways of implementing environmentally friendly aviation in the segment of regional transportation is the use of fundamentally new power plants based on hybrid-electric technology. The practical result of the implementation of such technology can be the development of a hybrid turbo-electric power plant (Hybrid Turbo-Electrical Propulsion - HTEP) using fuel cells (PE) powered by liquid hydrogen. In comparison with the well-known ATR-72-600 regional passenger aircraft, the CO2 emissions of the prospect aircraft with HTEP will be at least 30% lower.

This work is aimed at the preliminary design development of a regional passenger aircraft, which takes into account formation of the top-level requirements for the aircraft and systems, the peculiarities of using HTEP with fuel cells, as well as determining the characteristics of the aircraft taking into account the requirements of layout, aviation safety and operation.

Based on the results of the work, it is planned to obtain justification for the choice of aerodynamic layout and calculation of the energy balance for the design of a regional aircraft.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-3

Design of a Hybrid Turbo-Electric Propulsion (HTEP) with Fuel Cells for a Regional Aircraft with 80 Seats – Demonstration and Verification of some Solutions for HTEP.

Maksym Kirichkov*, Sergiy Dmytriyev, Oleksandr Yelanskiy, Igor Kravchenko [1]

[1] Ivchenko - Progress -, Zaporizhzhia, Ukraine

Key Words: *Hybrid turboelectric propulsion, regional aircraft, hydrogen, hybridization level, critical technologies*

This paper is based on the project EFACA (Environmentally Friendly Aviation for All Classes of Aircraft) that is supported by the European Union in the period 2023-2026.

The study is based on the idea that the image of a hybrid turbine-electric system with "fuel cells" for a regional aircraft with a capacity of up to 80 seats and a range of up to 1000 km will be determined by the type of energy supply (hydrocarbon or hydrogen fuel) to the gas turbine and electric engines.

The use of a sequential approach to the definition of the HTEP design includes the initial definition of the top-level requirements for the HTEP and the aircraft, the evaluation of conceptual schemes for the HTEP power supply, the preliminary formation of HTEP component designs, the further development of fuel supply, storage, fuel conditioning, and power distribution and manage systems.

In the next step, according to the selected evaluation criteria, the optimal scheme will be determined to provide energy to the HTEP with fuel cells. Taking into account the influence of the level of hybridisation of the propulsion system on its configuration, as well as a comparison of the hybrid-electric aircraft with its existing analogue (ATR 72), a more accurate assessment of the performance of the regional aircraft will be made.

The performance of the propulsion concept, which consists of an air propeller driven by a gearbox combining turbine and electric motor drives, will be assessed in the first TRL3 level demonstration bench test.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-4

Review of Liquid Hydrogen Tanks for Short- and Medium-Haul Aircraft

Daniel Terlizzi[1,*], Abdullah Bamoshmoosh[2] and Gianluca Valenti[3]

Department of Energy, Politecnico di Milano – Via Lambruschini 4A, 20156 Milan, Italy
[1]daniel.terlizzi@polimi.it
[2] abdullah.bamoshmoosh@polimi.it
[3] gianluca.valenti@polimi.it

Key Words: *Liquid Hydrogen, Hydrogen Storage, Cryogenic Tanks, Sustainable Aviation, Short- and Medium-haul Flights*

Hydrogen is increasingly seen as a key energy carrier in transitioning towards a decarbonized global energy system. In particular, hydrogen stands as a pivotal factor in fostering environmentally friendly aviation. The storage system is essential to utilize hydrogen for propulsion in commercial aircrafts. This review concentrates on cryogenic tanks for physically storing hydrogen in liquid form. The main challenge of integrating large liquid hydrogen tanks into the aircraft is the low volumetric density of the fuel. When compared to conventional fuels, hydrogen offers only a quarter of the energy content per unit volume, and three times the energy content per unit mass. Effective storage solutions are crucial to aircraft usefulness, since liquid hydrogen typically requires voluminous, heavy, and well-insulated tanks. Literature highlights the dependence of the optimal solution on the aircraft's range category.

This research focuses on the review of hydrogen fuel containment systems in short- and medium-haul flights. The objective of this analysis aims to spotlight the design intricacies involving key parameters such as diverse tank structures, layouts, shapes, venting pressures, and insulation strategies.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-5

Cooling Design and Refrigerant Selection of an Aviation PEM Fuel Cell System

Xin Gao[1,*], Arne Schweinitz[2], Michael Heere[3]

Inst. of Internal Combustion Engines and Fuel Cells, TU Braunschweig,
Hermann-Blenk-Straße 42, 38108 Braunschweig, Germany, https://www.tu-braunschweig.de/ivb
[1] xin.gao@tu-braunschweig.de,
[2] arne.schweinitz@tu-braunschweig.de,
[3] m.heere@tu-braunschweig.de

Key Words: *PEM Fuel Cells, PCHP cooling, Test rig*

Electromobility and especially air mobility is increasing its share among all transportation sectors. Along the process, hydrogen-electric propulsion relying on polymer electrolyte membrane (PEM) fuel cells as a most promising and key disruptive solution, is gradually stretching its popularity among aircraft belonged to more segments. However, in the trend, aviation still put stringent requirements on the 1) efficiency, 2) specific power, 3) specific energy, and 4) reliability, of these fuel cell systems.

Effective cooling could be a main contributor to all the requirements. Based on our previous simulation work proposing a novel cooling solution, namely PCHP cooling, in our current project EFACA (Environmentally Friendly Aviation for all Classes of Aircraft), we are building a test rig demonstrating the cooling concept. Currently, we are in the stage of detailing the design of the test rig including the selection of the major component, especially the phase-change refrigerant and deciding their optimal operating settings.

For these purposes, we are taking two paths, 1) thermodynamic modeling approach, and 2) industrial practices, i.e., engineering know-hows and market availability. In this presentation, we are going to show our findings and decisions so far along the progress of building such test rig.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-6

Promising Technologies to Reduce Aviation Noise at Airports

Andrzej Chyla[1], M. Bukala[1], K. Kazhan[2], V. Makarenko[2], V. Tokarev[2], Oleksandr Zaporozhets[3,*]

[1] Noise ACH, Warsaw, Poland, andrzej.chyla@gmail.com
[2] National Aviation University, Kyiv, Ukraine, kazhank@gmail.com
[3] ILOT - Institute of Aviation, Warsaw, Poland, zaporozhets.oleksandr@gmail.com

Key Words: *Aviation noise, reduction technologies, low noise designs, noise abatement procedures (NAPs)*

With highest priority of climate change in near future the aircraft noise in/around airports is still considered as important but with the priority global GHG emissions. Balanced approach realized during last few decades by ICAO provided the huge reduction of noise exposure around the airports worldwide and a number of impacted people by noise. Aviation sector is expected to triple the air traffic (taking in mind only the development of usual air traffic without the contribution of the new Urban Air Mobility and other Advanced Air Mobility approaches) through 2050 and the benefits of balanced approach must not be diminished. It means that noise reduction technologies will develop in a same way as during previous decades as for new designs of the aircraft so as for new low-noise operation procedures. If to look at new supersonic aircraft future technologies are also considered in mostly traditional development of noise reduction in engines and airframes (as considered now by EU SENECA project).

The EU concept of environmentally friendly aviation by 2050 (compared to 2000) provides for the use of new technologies for aircraft designs still in operation: to reduce the perceived aircraft noise by 65%. Achieving these goals requires evolutionary changes in aircraft design, improved aerodynamic configurations and the use of innovative engines. Taking in mind the highest priority for climate change goals globally and in particular for aviation sector the revolutionary aircraft designs are subject for noise control also. The existing taxonomy of noise sources and roadmaps for their reduction should be widened to cover the new designs and technologies.

The acoustic performances of radically new innovative aircraft concepts should be in line with evolutionary achievements. The electrification of aircraft propulsion promises a significant reduction of aviation emissions and progress toward of strategic goals achievement. But their main noise sources like the propellers and airframe dominant sources still need for further improvements to reach the EU strategic goal in perceptive noise. The recommendation for EU EFACA project relates to the implementation of hybrid electric propulsion technology (combination a gas turbine and electric engines) for propeller-driven regional aircraft must be accompanied with appropriate noise reduction efforts. Principles of aircraft hybrid electrification should be enough not only for necessary emission reduction of GHGs by regional aircraft in flight, but the goals in aircraft noise reduction should be reached.

An important element in the reduction of aircraft noise emissions around the airports is the use of optimal flight procedures – noise abatement procedures (NAPs). In combination with the planned changes in design technologies, the existing guidelines will need to be revised for NAPS.

Acknowledgement:

Acknowledgement to the EU Horizon Europe EFACA project No 101056866 “Environmentally Friendly Aviation for all Classes of Aircraft” and the SENECA project No 101006742 “((LTO) noiSe and EmissioNs of supErsoniC Aircraft)“.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-7

Promising Technologies to Reduce Global and Local Aviation Emissions

K. Synylo[1], A. Krupko[1], S. Karpenko[1], Oleksandr Zaporozhets[2,*]

[1] National Aviation University, Kyiv, Ukraine, kazhank@gmail.com
[2] ILOT - Institute of Aviation, Warsaw, Poland, zaporozhets.oleksandr@gmail.com

Key words: *Aviation emissions, local air quality (LAQ), greenhouse gas (GHG), climate change, fuel burn, sustainable aviation fuel (SAF), market-based measures (MBM)*

Climate change is the megatrend that will have the biggest impact on the development of sustainable air transportation in near future. Aviation sector is expected to triple its proportional share (taking in mind only the development of usual air traffic without the contribution of the new Urban Air Mobility and other Advanced Air Mobility approaches) of a Paris compatible 1.5°C budget. ICAO basket of measures to keep the temperature change under this limit, including aircraft technology (up to 25%) and operation improvement (up to 9%) for fuel burn reduction by engines and new revolutionary designs of the aircraft, deployment of sustainable alternative fuels (over 40% of fuel burn reduction), market-based measures as pushing system for quicker and more efficient implementation of the first three, etc. Pioneering sustainable technology is allowing the civil aviation sector to embrace the next generation of aviation through electrification and alternative fuels including hydrogen. If to look at new supersonic aircraft future technologies are considered in mostly traditional development of emission reduction in combination with SAF for fuel burn by their engines (as considered now by EU SENECA project).

Radically new innovative aircraft concepts are necessary for implementation in aircraft design, which efficiency in emission reduction should be much higher than for current evolutionary concepts. The electrification of aircraft propulsion promises a significant reduction of aviation emissions and progress toward the strategic goals achievement. The first recommendation for EU EFACA project relates to the implementation of hybrid electric propulsion technology for propeller-driven regional aircraft. Principles of aircraft hybrid electrification should be enough for necessary emission reduction of GHGs by regional aircraft in flight, so as the goals in aircraft noise and local air quality should be reached.

Hydrogen fuel, which is burnt in combustion chambers of the engines directly (instead of usual fossil fuel or/and SAF from renewable resources), is the crucial potential technology for eliminating aircraft GHG emissions for the most popular groups of aircraft – short/medium range and long-range aircraft (their contribution in aviation sector is over the 90 %). Airbus is evaluating several hydrogen approaches for future aircraft designs (ZEROe), which include “direct hydrogen” and “hydrogen fuel cells”. With hydrogen as a fuel, either in combustion engines or used in a fuel cell, there are no in-flight CO_2 emissions whatsoever. The second recommendation for EU EFACA project relates to the implementation of hydrogen fuel cells and liquid hydrogen fuel system for propeller-driven regional aircraft and jet airliner correspondingly.

Acknowledgement:

Acknowledgement to the EU Horizon Europe EFACA project No 101056866 “Environmentally Friendly Aviation for all Classes of Aircraft” and the SENECA project No 101006742 “((LTO) noiSe and EmissioNs of supErsoniC Aircraft)“.

STS 245-8

Prospects of Novel Technologies for SAF Production

Renata Adami[1,2,*], Patrizia Lamberti[2,3], Anna Yakovlieva[4], Sergii Boichenko[4], Maria Sarno[1,2] and Vincenzo Tucci[2,3]

[1] University of Salerno, Department of Physics "E.R. Caianiello", via Giovanni Paolo II 132, Fisciano (SA), Italy, radami@unisa.it, https://www.df.unisa.it/

[2] Interdepartmental Centre NANO_MATES (Research Centre for NANOMAterials and nanoTEchnology), via Giovanni Paolo II 132, Fisciano (SA), Italy, https://www.nanomates.unisa.it/

[3] University of Salerno, Department of Information, Electrical Engineering and Applied Mathematics, via Giovanni Paolo II 132, Fisciano (SA), Italy, https://www.diem.unisa.it/

[4] Igor Sikorsky Kyiv Polytechnic Institute, Institute of Energy Safety and Energy Management, Borshchahivska str, 115, Kyiv, Ukraine, https://www.kpi.ua/en/

Key words: *Sustainable Aviation Fuels, ASTM, Advanced Biofuels, e-fuels, e-kerosene*

For very large aircraft and very long distances, low-carbon alternative fuels are at present the only viable replacement in near term for fossil fuels, contributing towards sustainability targets before electric and hydrogen technologies will become mature. Since the interest on Sustainable Aviation Fuels (SAF) is growing, more pathways will be introduced and approved in the near future.

The scientific research is mainly focusing on the production of advanced biofuels and e-kerosene. Biofuels from biomass or waste (cooking oils, fats) and advanced biofuels that are synthesized from solid feedstock, biomass like crops, or algae. E-fuels or e-kerosene are synthetic fuels or synfuels, synthesized from hydrogen and carbon dioxide taken from industrial, biomass or direct-air capture.

For the advanced biofuels, beyond existing modern technologies for SAFs production, novel pathways are proposed, mainly related to the conversion and fermentation of feedstocks to alcohol and then biofuels. The main techniques are: Alcohol to Jet Synthetic Kerosene with Aromatics (ATJ-SKA), approved by ASTM (American Society for Testing and Materials) in 2023, and the novel technologies L-ETH-Jet, Biochemical conversion of lignocellulose to ethanol process; SYN-FER-J, Gasification, syngas fermentation to ethanol process; S-ETH-J, Sugarcane juice to ethanol by sucrose fermentation process.

E-fuels are produced using renewable energy for fuel synthesis. New technologies for e-fuels production, like power-to-liquid process are promising for reducing greenhouse gas emissions in aviation. They are not yet available on the market, but there is a growing interest from various companies in establishing production facilities. Moreover, a growing enthusiasm for the electrochemical route has potential to improve e-kerosene production.

References:

[1] R. Tiwari, R. Mishra, A. Choubey, S. Kumar, A.E. Atabani, I. Anjum Badruddin, T.M. Yunus Khan. Environmental and economic issues for renewable production of bio-jet fuel: A global prospective. [Online] 2023. https://doi.org/10.1016/j.fuel.2022.125978.

[2] Yakovlieva, A., Boichenko, S., Boshkov, V., Korba, L., & Hocko, M. Experimental study of physical-chemical properties of advanced alcohol-to-jet fuels. [Online] 2023. https://doi.org/10.3846/aviation.2023.18564.

[3] Martinez-Valencia, L., Supply chain configuration of sustainable aviation fuel: Review, challenges, and pathways for including environmental and social benefits. [Online] 2021. https://doi.org/10.1016/j.rser.2021.111680.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 245-9

Market Prospects for a Hybrid Turbo-Electric Propulsion (HTEP) with Fuel Cells for a Regional Aircraft with 80 Seats

Maria Zamarreño Suarez*, Raquel Delgado-Aguilera Jurado, Francisco Perez Moreno, Victor Fernando Gomez Comendador, Rosa Arnaldo Valdés[1]

[1] Universidad Politecnica de Madrid, Plaza Cardenal Cisneros, 3 Madrid 28040, Spain
Rosamaria.arnaldo@upm.es, www.upm.es

Key Words: *Regional aircraft, hybrid turbine-electric propeller, fuel cells, batteries, potential market size.*

The classification of commercial airplanes typically includes three categories: single-aisle (narrow-body), twin-aisle (wide-body), and regional aircraft, the smallest variant. This classification aids in understanding sizes, capacities, and integrating hybrid technologies. Aircraft with electric and hybrid propulsion systems are expected to impact the regional market, potentially replacing turboprop models but not conventional jets entirely.

Before COVID-19 emerged, regional aviation experienced significant growth, comprising over 12% of global available seat kilometres (ASK). However, the pandemic's aftermath profoundly impacted global economic progress and air travel, leading to a gradual recovery. Projections indicate that global passenger traffic, measured in revenue passenger kilometres (RPK), will return to pre-pandemic 2019 levels by 2024. This recovery is driven by pandemic recovery efforts, geopolitical factors, and shifts within the aviation industry. Forecasts suggest that global RPKs will maintain a steady annual growth rate of 3.2% until 2042. The translation of this growth into the regional aviation market, along with evolving dynamics, will significantly influence the adoption and success of innovative hybrid aviation technologies.

The EFACA Project, funded by the European Union from 2023 to 2026, focuses on developing a hybrid turbine-electric propeller (HTEP) aircraft system integrated with fuel cells for regional aircraft with up to 80 seats and 1000 km ranges. The project outlines top-level requirements, evaluates power supply frameworks, develops component designs, and enhances fuel supply, storage, conditioning, and power distribution systems. This paper explores the potential market for the HTEP aircraft design, analyzing affected markets, including hybrid and fuel cell-powered aircraft, and the regional aircraft sector. It scrutinizes market shares within regional and short-haul routes pre- and post-pandemic, offering insights crucial for refining top-level aircraft requirements, particularly flight range. The paper is organized as follows: Section 1 corresponds to introduction. Section 2 discusses the technological attributes of the proposed hybrid electrical fuel cell aircraft concept; Section 3 delves into the prospects for various aircraft markets, aiming to identify the most viable segments for the hybrid aircraft; Section 4 presents an analysis of market shares for regional and short-haul routes pre- and post-pandemic to estimate potential market size; and finally, Section 5 offers concluding remarks.

Acknowledgement:

Acknowledgement to the EU Horizon Europe EFACA project No 101056866 “Environmentally Friendly Aviation for all Classes of Aircraft”.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 247

Disruptive Aircraft's Wing Configurations towards Greening of Aviation

Chairs: Marianna Braza[1] and Yannick Hoarau[2]

[1] Institut de Mécanique des Fluides de Toulouse, IMFT, UMR 5502 CNRS-INPT-UT3 UMR, Allée du prof. Camille Soula, 31400 Toulouse, France, marianna.braza@imft.fr

[2] ICUBE - University of Strasbourg, ICUBE UMR 7357, Laboratoire des Sciences de l'Ingénieur, de l'Informatique et de l'Imagerie, 67081 Strasbourg, France, hoarau@unistra.fr

Session Abstract

Key Words: *Smart Morphing and Sensing, Wing Design, High-Fidelity Numerical Simulations, Aerodynamic Performance, Emissions Reduction*

The present STS at ECCOMAS 2024 will include five contributions concerning novel wing morphing, able to drastically increase the aerodynamic performances leading to a considerable fuel's consumption decrease and noise sources reduction**.** Emphasis will be attributed in the efficiency of multiscale electrical actuations with increased DoF over strategic areas of the lifting structures. The presentations analyse the morphing effects on the fluid-structure interaction, beneficially manipulating the surrounding turbulence towards drag reduction, increase of lift and noise sources attenuation. The new morphing designs ensure a considerable energy decrease for the propulsion, beneficial for all sources of renewal energy.

These studies are a continuation from the EU-funded Horizon 2020research project N° 723402 SMS, "Smart Morphing and Sensing for aeronautical configurations", https://cordis.europa.eu/project/id/723402 and www.smartwing.org/SMS/EU.

They are performed in the context of the HORIZON-EIC-2023-PATHFINDER Project N° 101129952 – BEALIVE, "Bioinspired Electroactive multiscale Aeronautical Live skin", https://cordis.europa.eu/project/id/101129952.

The presentations included in this STS analyse through High-Fidelity numerical approaches, the effects of spatial and temporal modulation of the actuation frequencies and amplitudes applied through novel smart actuators disposed in a distributed way on the "skin" of the lifting structure. These designs are able to produce optimal interfacial layers interacting with the coherent and chaotic turbulence structures and applying deformation of strategic parts of the wing.

The topic of this session prepares future wing design for aeronautics industrial applications aiming at saving energy and at reducing the pollution through these new, *multiple-degrees-of freedom morphing concepts*, enabling a considerable reduction of emissions, meeting the targets fixed by Flightpath 2050: Europe's Vision for Aviation [1].

[1] European Commission, DG MOVE/ DG RTD, Flightpath 2050: Europe's Vision for Aviation: Maintaining global leadership and serving society's needs, Publications Office, 2012, https://data.europa.eu/doi/10.2777/15458.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 247-1

Investigation of the Electroactive Morphing Effects on an A320 Prototype in Subsonic Regime at Reynolds Number of 1 Million

Jacques Abou Khalil[1,2,*], Mateus Carvalho[1,2,3], Clément Rouaix[2,3], Cédric Raibaudo[4], Gilles Harran[2], Sébastien Cazin[1], Moïse Marchall[2], Frédéric Bergame[2], Hervé Ayroles[2], Jean-François Rouchon[1], Marianna Braza[2]

[1] LAPLACE Laboratoire Plasma et Conversion d'Energie, UMR 5213 CNRS-INPT-UT3
[2] Institut de Mécanique des Fluides de Toulouse (IMFT) UMR 5502 CNRS-INPT-UT3
[3] Ontario Tech University, Oshawa, Canada
[4] Laboratoire PRISME, Université d'Orléans - INSA, Orléans, France
jacques.aboukhalil@toulouse-inp.fr

Key Words: *Morphing, Fluid-Structure Interaction, Control, Turbulence Modeling, Wing*

The present paper presents a new electroactive morphing concept studied by the IMFT and LAPLACE laboratories, able to drastically increase the aerodynamic performance. This concept, using a series of MFC (Macro-Fiber-Composite) piezoelectric actuators disposed along the trailing -edge area, as well as along specific rows in the region of 70% of the chord, has been investigated by High-Fidelity numerical simulations and refined physical experiments around the so-called "Reduced Scale" (RS) A320 wing prototype of 70 cm chord of the HORIZON-PATHFINDER Project N° 101129952 - BEALIVE - "Bioinspired Electroactive multiscale Aeronautical Live skin". The experiments and the simulations have been accomplished at 10° of incidence, Mach number of 0.06 and Reynolds number of 1 Million, corresponding to onset of take-off conditions. A specific emphasis has been attributed on the lift increase, drag reduction and on noise sources reduction by attenuation of harmful instabilities. The actuators motion and the slight deformation of the trailing-edge area has been numerically modelled by means of the Arbitrary Lagrangian-Eulerian (ALE) method with the CFD solver NSMB (Navier-Stokes Multi Block). Concerning the turbulence modelling, the Organised Eddy Simulation (OES) approach [1] has been employed. As demonstrated in our numerical and experimental studies, the Trailing-Edge (TE) vibration significantly increases the lift force in the order of 5%, while generation of Travelling Waves (TW) along the rear part of the suction side notably reduces drag by an order of 6%. These performances have been obtained through generation of smaller-scale vortices from the vibrations that interact with the existing coherent vortices in the shear layers (Kelvin-Helmholtz (KH)) and von Kármán (VK) vortices farther downstream, as well as with smaller chaotic turbulent vortices. The dynamic manipulation of these vortices produces an "*eddy-blocking*" effect, that leads to a considerable *thinning* of the shear layers and of the wake's width.

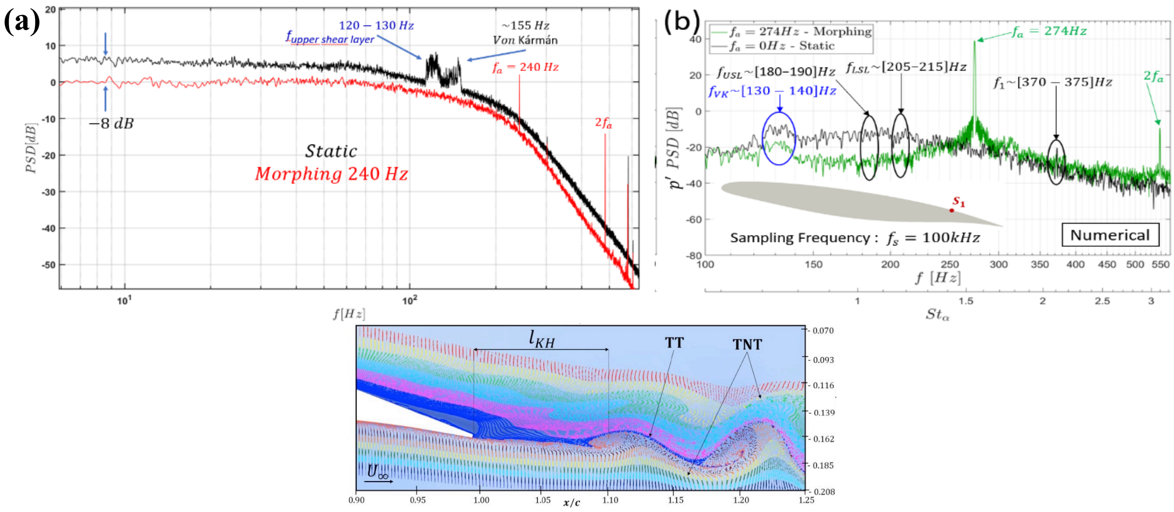


Figure 1: *Top: PSD of pressure fluctuations signal for the morphing case and static case for the pressure sensor S1 located at x/c=0.80. Re=10^6. Angle of incidence 10°. (a) experiments; (b) simulations. Bottom: zoom of the flow structure near the trailing edge and wake. TT: Turbulent-turbulent interface. TNT: Turbulent-non-turbulent.*

References

[1] R. Bourguet, M. Braza, G. Harran, R. El Akoury (2008) https://doi.org/10.1016/j.jfluidstructs.2008.07.004

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 247-2

Three-dimensional Numerical Simulations around an A320 Morphing Wing at High Reynolds Number

Clément Rouaix[1,2], Abderahmane Marouf [3], Yannick Hoarau[3], Mateus Carvalho[1,2], Horia Hangan[2], Marianna Braza[1]

[1] Institut de Mécanique des Fluides de Toulouse (IMFT), UMR 5502 CNRS-INPT-UT3
[2] Faculty of Engineering and Applied Science (FEAS), Ontario Tech University, Oshawa, Canada
[3] ICUBE Laboratoire des sciences de l'Ingénieur, de l'Informatique et de l'Imagerie, UMR 7357 CNRS-Univ. Strasbourg-ENGEES-INSA, France
clement.rouaix@imft.fr; clement.rouaix@ontariotechu.ca

Key words: *Morphing, wing, frequency modulation, wobulation, CFD, turbulence*

3D numerical simulations are performed around the Reduced Scale A320 wing prototype of the H2020 N°723402 EU project SMS: "Smart Morphing and Sensing for aeronautical configurations", https://cordis.europa.eu/project/id/723402/fr, www.smartwing.org/SMS/EU. The wing has 70cm chord and 56cm span and is investigated in the subsonic regime at Reynolds number 1 million and for 10° incidence using the Navier Stokes Multi-Block (NSMB) code. Bioinspired electroactive morphing is numerically investigated through vibration and slight deformation in the near trailing edge region, using the ALE method for mesh deformation and the Organised Eddy Simulation (OES) turbulence modelling approach, [1], sensitized to the coherent structures physically correct development, also including the DDES-OES case (Detached Eddy Simulation with embedded OES). Following a previous 2D numerical study [2], a linear time modulation of the trailing edge actuation frequency (wobulation) is carried out in 3D to efficiently detect optimal morphing frequencies providing aerodynamic performance increase.

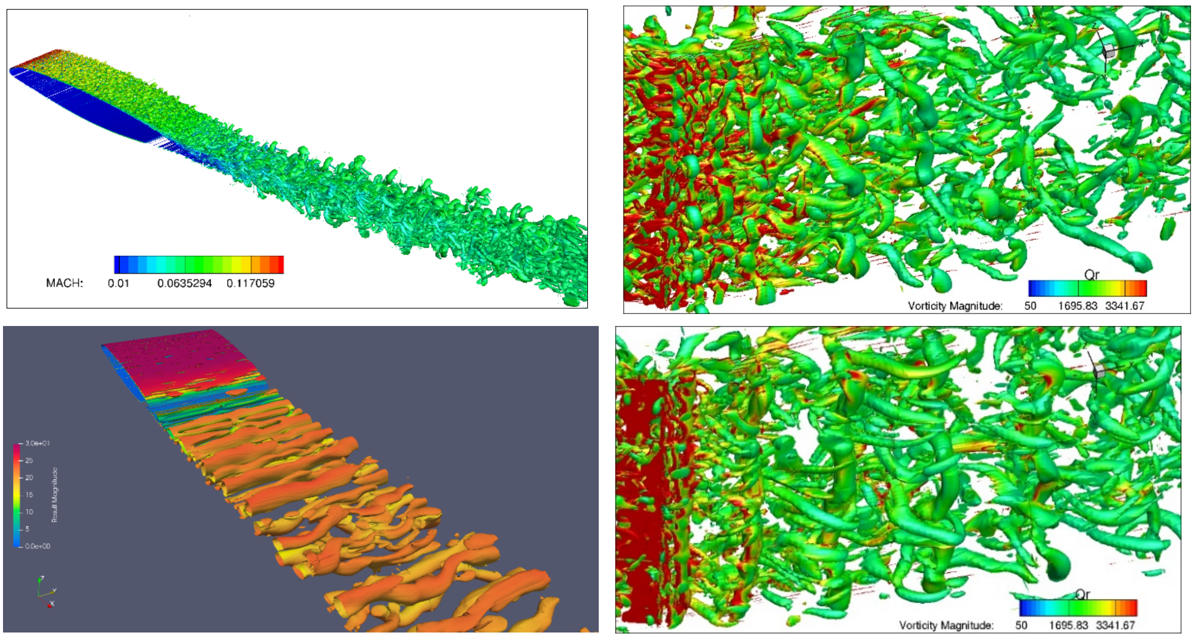


Figure 1: ***Top:*** Static (unactuated) case showing the separation area (blue) and the multitude of vortices including shear-layer and von Kármán ones, as well as the secondary instability along the span. ***Bottom-left:*** Morphing with linear variation versus time of the actuation frequency (wobulation) in the near-trailing edge region showing considerable attenuation of the secondary instability. ***Bottom-right:*** enhancement of a more orderly vortex structure than the static case with decrease of the spanwise undulation, shown through the Q criterion

References:

[1] R. Bourguet, M. Braza, G. Harran, R. El Akoury (2008), https://doi.org/10.1016/j.jfluidstructs.2008.07.004

[2] C. Rouaix, C. Jiménez-Navarro, M. Carvalho, C. Raibaudo, J. Abou-Khalil, A. Marouf, Y. Hoarau, G. Harran, J. C. R. Hunt, H. Hangan, J.F. Rouchon and M. Braza, https://doi.org/10.1016/j.jfluidstructs.2023.104016

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 247-3

Numerical Investigation of Electroactive Morphing Effects through Traveling Wave Actuation on an A320 Wing in Low Subsonic Regime and Reynolds Number of 1 Million

Rajaa El Akoury[1,*], Abderahmane Marouf [2], Jacques Abou Khalil [1,3], Yannick Hoarau[2], Jean-Francois Rouchon[3] and Marianna Braza[1]

[1] Institut de Mécanique des Fluides de Toulouse (IMFT) UMR 5502 CNRS-INPT-UT3
[2] ICUBE Laboratoire des sciences de l'Ingénieur, de l'Informatique et de l'Imagerie, UMR 7357 CNRS-Univ. Strasbourg-ENGEES-INSA, France
[3] LAPLACE Laboratoire Plasma et Conversion d'Energie, UMR 5213 CNRS-INPT-UT3
rajaa.elakoury@toulouse-inp.fr

Key words: *Morphing, Traveling Waves, wing, fluid-structure interaction, turbulence modeling*

A new bioinspired electroactive morphing concept through Traveling Waves (TW) applied along a rear part of the suction side of the "Reduced Scale (RS) A320 morphing wing prototype of the HORIZON-2023-PATHFINDER-Open-Project N° 101129952-BEALIVE – "Bioinspired Electroactive multiscale Aeronautical Live skin" has been numerically investigated in low subsonic regime (Mach number of 0.063), Reynolds number of 1 Million and incidence of 10°, corresponding to take-off conditions. The TW are implemented over an optimised zone of the wing's suction side (Fig.1). The effects of this morphing concept, (examined in parallel through experimental studies by IMFT & LAPLACE Laboratories) are investigated in respect of aerodynamic performance increase. The simulations used the Navier Stokes Multi-Block (NSMB) code, including the Arbitrary Lagrangian Eulerian (ALE) methodology for dynamic grid deformation, and the Organised Eddy Simulation (OES) turbulence modelling approach, [1,2]. The RS prototype has a 70 cm chord and span of 56 cm, operating at an incidence angle of 10°. A boundary layer separation occurs in the static (non-morphing) case, at around 80% of the chord. The significant shear downstream results in the formation of Kelvin-Helmholtz (KH) vortices, and farther downstream, to von Kármán (VK) vortex shedding. A very large parametric investigation has been carried out, involving thousands of combinations of the following parameters to analyse the influence on the aerodynamic forces and study optimal regimes: wavelength (λ), frequency (f), amplitude (a), and the actuation zone location, $[x_0/C, x_f/C]$. The oscillations created by the waves interact with the separation and the shear layers and promote a more organized flow structures with *a significant reduction of the spanwise secondary instability.* This is obtained through manipulation of the detrimental effects of surrounding turbulence through this morping concept and producing significant impacting *feedback effects* on the upstream wall pressure distribution, leading to increased performances. For optimal sets of the TW parameters, *an increase in the lift-to-drag performance reaching **9%*** has been obtained The optimal parameters are used in the experimental studies carried out in the BEALIVE project, where the traveling waves can be materialised through novel piezo-actuators.

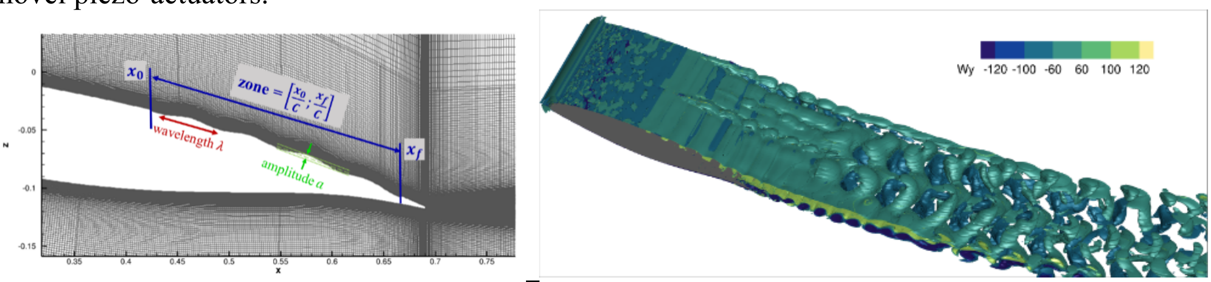


Figure 1: Traveling wave morphing parameters (left), vortical structures manipulated through Travelling Wave morphing ($f = 300Hz, a = 1mm, \lambda = 6cm, Zone = [0.2 - 0.7]m$)

References

[1] M. Braza, R. Perrin, Y. Hoarau (2006) https://doi.org/10.1016/j.jfluidstructs.2006.04.021
[2] R. Bourguet, M. Braza, G. Harran, R. El Akoury (2008) https://doi.org/10.1016/j.jfluidstructs.2008.07.004

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 247-4

Investigation of the Aerodynamic Performance Increase in Transonic Flow over an A320 Morphing Airfoil by Numerical Simulation at High Reynolds Number

Jacques Abou-Khalil[3,1], César Jimenez-Navarro[1], Aberahmane Marouf[1,2], Rajaa El Akoury[1], Yannick Hoarau[2], J.F. Rouchon[3] and Marianna Braza[1]

[1] Institut de Mécanique des Fluides de Toulouse (IMFT) UMR 5502 CNRS-INPT-UT3, Toulouse, France
jacques.aboukhalil@toulouse-inp.fr
[2] ICUBE Laboratoire des sciences de l'Ingénieur, de l'Informatique et de l'Imagerie, UMR 7357 CNRS-Univ. Strasbourg-ENGEES-INSA, Strasbourg, France
[3] LAPLACE Laboratoire Plasma et Conversion d'Energie, UMR 5213 CNRS-INPT-UT3, Toulouse

Key Words: *Morphing, transonic regime, Traveling Waves, wing, fluid-structure interaction*

The present study focuses on a morphing A320 wing prototype with a 70 cm chord length, incidence of 1.8°, upstream Mach number of 0.78 and Reynolds number 4.5 x 10^6. The study is part of the HORIZON-2023-PATHFINDER-Open-Project N° 101129952-BEALIVE - "Bioinspired Electroactive multiscale Aeronautical Live skin". This study aims at exploring actuation surfaces beyond the most downstream position of the shock in respect of buffet, creating a "live-skin" surface through a Traveling Wave (TW) approach (Figure 1). A large numerical parametric study is conducted to identify optimal frequency, amplitude and wave-length ranges of the TW in respect of aerodynamic performance increase. The grid deformation due to the TW has been handled by the Arbitrary Lagrangian-Eulerian (ALE) method in the CFD solver NSMB (Navier-Stokes Multi Block). The Organised Eddy Simulation (OES) approach has been used for the turbulence modelling, sensitized to coherent structures physical development capturing. The study revealed the development of buffet instability at 1.8° angle of incidence concerning the unactuated (static) case. Furthermore, the research highlighted a significant feedback effect from the near wake unsteadiness towards the Shock Boundary Layer Interaction (SBLI) region and even upstream of it. By enhancing this effect through morphing that manipulates the downstream unsteadiness, (fig.1) specifically the Kelvin-Helmholtz and von Kármán vortices, the study demonstrated that this morphing effectively reduces the wake thickness and suppresses the buffet instability (fig.2) yielding a drag reduction in the order of 6% and a lift-to-drag increase of 2.5%.

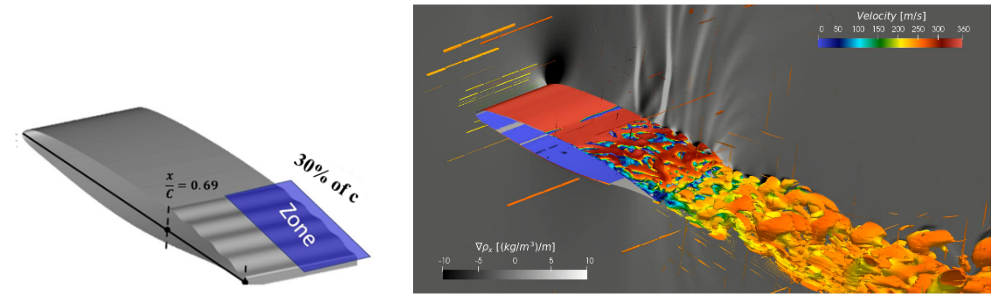


Figure 1: Left: Travelling Wave approach. Right: Illustration of the formation of KH vortices within the buffet cycle and their feedback effect together with the near wake unsteadiness, both affecting the upstream region.

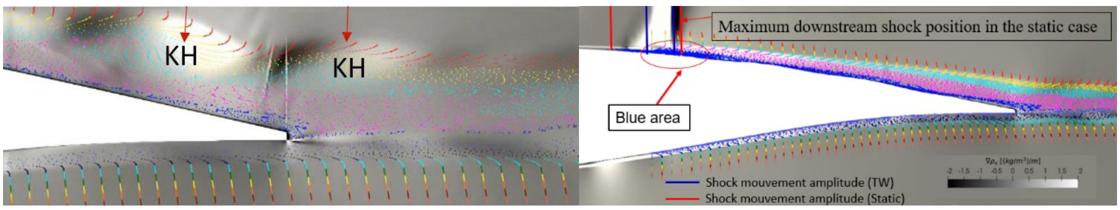


Figure 2: Left: static case: formation of KH vortices and thick wake. Right: TW case: reduction of the wake thickness and suppression of the shear layer instabilities by reducing the shock motion: difference between blue (morphing) and red (static) vertical lines.

References

[1] M. Braza, R. Perrin, Y. Hoarau (2006) https://doi.org/10.1016/j.jfluidstructs.2006.04.021
[2] R. Bourguet, M. Braza, G. Harran, R. El Akoury (2008) https://doi.org/10.1016/j.jfluidstructs.2008.07.004

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 247-5

Reduced Order Modelling for Aerodynamic Turbulent Flow around an A320 Morphing Wing Prototype

Abderahmane Marouf[1], Rajaa El Akoury[2], Marianna Braza[2], and Yannick Hoarau[1,*]

[1] ICUBE Laboratoire des sciences de l'Ingénieur, de l'Informatique et de l'Imagerie, UMR 7357 CNRS-Univ. Strasbourg-ENGEES-INSA, Strasbourg, France, amarouf@unistra.fr

[2] Institut de Mécanique des Fluides de Toulouse (IMFT) UMR 5502 CNRS-INPT-UT3, Toulouse, France, marianna.braza@imft.fr

Key Words: *Morphing, POD, Machine Learning, Turbulent, Wing, Simulation*

In the context of the European project HORIZON-2023-PATHFINDER-Open-Project N° 101129952-BEALIVE- "Bioinspired Electroactive multiscale Aeronautical Live skin" and the national French research project ANR-EMBIA, a Reduced Order Modelling (ROM) is proposed for turbulent flows surrounding morphing wings. The Reduced Scale (RS) A320 wing prototype of the BEALIVE project has been considered at a Reynolds number of 10^6 and incidence of 10 degrees in the low subsonic regime (Mach number in the order of 0.06). The wing is deformed during time using a Traveling Wave (TW) concept, trailing-edge vibrations [1,2], or cambering [3]. This leads to a displacement of the coordinates as a function of time and requires a mesh deformation. As consequence, large parametric studies with various parameters require time and resources. To tackle this issue, a ROM is developed based on the Proper Orthogonal Decomposition (POD) approach and Machine Learning. The ROM is constructed using an enriched POD of multiple simulations. An interpolation is conducted using a trained Machine Learning algorithm to improve the prediction. The new model not only accounts mean solutions efficiently but predicts transient dynamics like boundary layer separation, shear layer, and main instabilities in the near wake associated with coherent structures development. This is achieved while maintaining and capturing correct energy distribution of turbulent structures and considering smaller energy chaotic structures by using spatial modes (Figure 1).

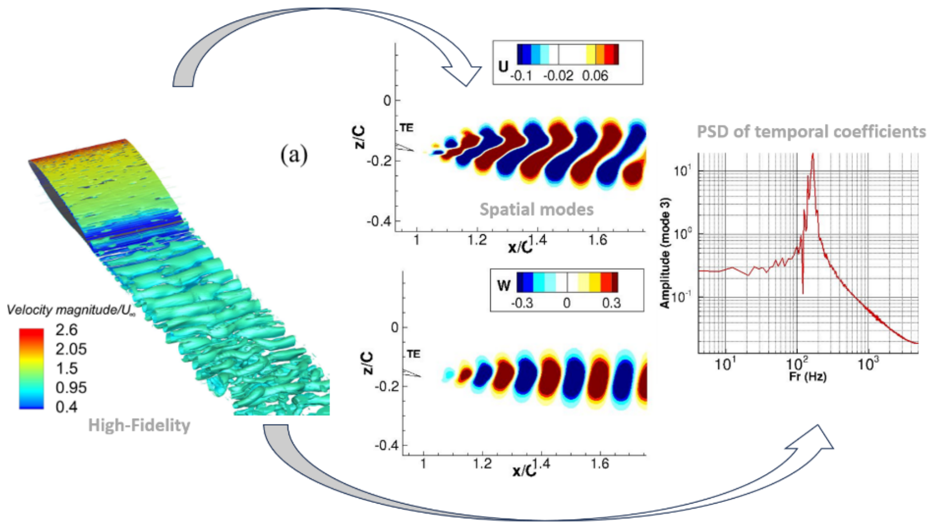


Figure 1: High-Fidelity simulation with POD high energy modes and spectra of their temporal coefficients as well as POD shape modes in the near wake.

References:

[1] A. Marouf, Y. Hoarau, J. B. Vos, D. Charbonnier, Y. T. Bmegaptche Tekap and M. Braza. Evaluation of the aerodynamic performance increase thanks to a morphing A320 wing with high-lift flap by means of CFD Hi-Fi approaches. AIAA Aviation 2019 Forum. https://doi.org/10.2514/6.2019-2912

[2] Marouf, A. et al. (2022). Smart Morphing and Sensing for the Wings of the Future. In: Knoerzer, D., Periaux, J., Tuovinen, T. (eds) Advances in Computational Methods and Technologies in Aeronautics and Industry. Computational Methods in Applied Sciences, vol 57. Springer, Cham. https://doi.org/10.1007/978-3-031-12019-0_2

[3] Y. T. Bmegaptche Tekap, A. Giraud, G. Jodin, C. Nadal, A. Marouf, I. Gortazar Martinez, D. Harribey, B. Nogarede, J-F. Rouchon and M. Braza. "Design of a Large-Scale High-Lift Morphing A320 Wing Based on Electro-Mechanical Actuators and Shape Memory Alloys,". AIAA Aviation 2019 Forum. https://doi.org/10.2514/6.2019-2908

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 247-6

Airfoil Shape Control using SMA Actuators

Michalis Fragiadakis[1,*], Nikolaos Simiriotis[2], Jean-Francois Rouchon[3] and Marianna Braza[2]

[1] National Technical University of Athens (NTUA), 9 Iroon Polytechniou 15780, Zografou Campus, Athens, Greece, mfrag@mail.ntua.gr

[2] Institut de Mécanique des Fluides de Toulouse (IMFT), UMR 5502 CNRS-INPT-UPS, 2 Allée du Professeur Camille Soula, 31400 Toulouse, France
nikolaos.simiriotis@toulouse-inp.fr, marianna.braza@imft.fr

[3] Laboratoire Plasma et Conversion d'Energie (LAPLACE), UMR 5213 CNRS-INPT, 2 rue Charles Camichel, 31071 Toulouse, France, francois.rouchon@laplace.univ-tlse.fr

Key Words: *SMA, Smart materials, Morphing, Shape control, Optimization*

The paper introduces an innovative methodology for efficiently designing intelligent deformable aeronautical configurations capable of achieving predefined target shapes through the strategic adjustment of Shape Memory Alloy (SMA) actuators. SMA-based actuation plays a pivotal role in morphing concepts widely utilized in aeronautics, contributing to improved aerodynamic performance by dynamically altering wing geometry.

The study presents a novel and robust algorithm specifically crafted to predict the nonlinear response within the SMA-structure interaction, further enhanced by its integration with an optimization method. This combined approach facilitates the accurate prediction of optimal structural and operational parameters aligned with the desired shapes for the controlled configuration. The design methodology, outlined in this research, meticulously selects key design parameters — such as the actuators' placement and operational temperature — tailored to specific loading conditions. Different case studies are considered in order to showcase the implementation of the proposed approach.

STS 248

Aerodynamic Shape Optimisation for Reducing Aircraft Emission and Increasing Wind Turbine Efficiency

Chairs: Ning Qin[1] and Chunling Zhu[2]

[1] Department of Mechanical Engineering, The University of Sheffield, Sheffield, S1 3JD, South Yorkshire, United Kingdom, n.qin@sheffield.ac.uk

[2] Nanjing University of Aeronautics and Astronautics - NUAA, Nanjing, China

Session Abstract

Key Words: *Aerodynamic shape optimisation, novel aircraft configurations, offshore wind turbines, adjoint methods*

This session will include contributions from researchers working on advanced aerodynamic simulation and shape optimisation, such as hybrid RANS/LES, and shape design optimisation, such adjoint based methods, for applications from the design of very large off-shore wind turbine blades to potential future large transport aircraft, such as the blended wing body and the truss-braced wing aircraft.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 248-1

Meso-Microscale Coupled Modelling of Wind Resource over Complex Terrain Wind Farm

Yilei Song[1], Guolin Ma[1], Ning Zhao[1]* and Linlin Tian[1]

[1] Jiangsu Key Laboratory of Hi-Tech Research for Wind Turbine Design,
Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China
zhaoam@nuaa.edu.cn (N. Zhao)

Key Words: *WRF, LES, Mesoscale models, Complex terrain, Atmospheric stability*

Due to the rapid deployment of wind energy, most newly constructed wind turbines are built on varied terrains to take advantage of topographic flow enhancement. During this process, other terrain-induced flow phenomena in the atmospheric boundary layer (ABL) would affect the performance of wind turbine as well as their wake characteristics. Thus, prior to wind farm construction, the wind resource assessment and wind turbine wake behavior evaluation in the complex terrain are become increasingly crucial for wind energy projects. However, knowledge on these issues is still limited.

In the complex wind farm, a highly unsteady and multi-scale coupling phenomenon exists, with the largest vortices being on the scale of kilometers while the smallest vortices are on the scale of millimeters. Under such circumstances, numerical simulation is a powerful tool to help understand the temporal-spatial characteristics and mechanism behind the complex wind phenomenon. Thus, the coupling of mesoscale and microscale simulation method, which is designed to predict the spatial changes in the multi-scale turbulent wind flow, becomes a promising way in the wind energy research.

In this work, a meso-microscale coupling methodology is developed and used to investigate wind characteristics under varying atmospheric stabilities over complex terrain. In which, the Weather Research and Forecast (WRF) model is used to mesoscale meteorology simulation, meanwhile the Large-Eddy Simulation (LES) method is employed to capture transient turbulent flow structures, which are important features of the ABL that interact with turbine wakes. The coupling methodology uses time-varying flow outputs from the mesoscale WRF model as boundaries for the microscale LES model. With Askervein hill and Blound island as test cases, the coupled simulation performs well at reproducing the vertical profiles of averaged wind speed. With atmosphere gets more stable, shear patterns get more pronounced. In the contrast, surface layer is more well-mixed under convective stabilities.

References

[1] P. Durán, C. Meiβner, P. Casso. A new meso-microscale coupled modelling framework for wind resource assessment: A validation study. Renewable Energy, 160: 538:554, 2020.

[2] S. Heinz, Theory-Based Mesoscale to Microscale Coupling for Wind Energy Applications. Applied Mathematical Modelling, 98: 563-575, 2021.

[3] A. Wise, J. Neher, R. Arthur, et al. Meso- to micro-scale modeling of atmospheric stability effects on wind turbine wake behavior in complex terrain. Wind Energy Science. 7: 367-386, 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 248-2

Manifold-Guided Multi-Objective Optimization for Supersonic Aircraft Shape Design

Zhili Tang[1], Fan Cao[1], Caicheng Zhu[1], Xin Zhao[1]*

[1] College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics - NUAA, Nanjing 210016, China

Key Words: *Multi/Many-objective optimization, Adjoint optimization, Supersonic civil aircraft, Pareto front manifold, Manifold sampling*

The aerodynamic design of supersonic aircraft necessitates the comprehensive consideration of multiple conflicting design objectives and multi-disciplinary characteristics. However, the efficiency of heuristic algorithms in solving multi-objective aerodynamic problems significantly decreases with the increase in the number of design variables and objectives. Although gradient-based adjoint optimization methods can eliminate the constraints of design variables on aerodynamic optimization and have high optimization efficiency, they are prone to getting stuck in local optima and are difficult to apply in multi-objective optimization. To overcome these shortcomings, a manifold-guided multi-objective gradient-based algorithm (MG-MOGBA) is proposed. This algorithm enhances the optimization efficiency of the algorithm by coupling the multi-objective gradient operator with the predicted Pareto front (PF) manifold structure to guide the population search direction and reduce the search space. A multi-objective aerodynamic optimization framework based on the discrete adjoint method is developed to automate the aerodynamic optimization process. Test functions and airfoil optimization results confirm that the proposed algorithm achieves superior results with fewer iterations and is more efficient than heuristic algorithms. The four-objective supersonic civil aircraft example demonstrates that the algorithm can effectively handle high-dimensional multi-objective aerodynamic optimization problems. A potential equilibrium solution increased the cruise factor by 17.5% and 12.8% under supersonic and transonic conditions respectively, reduced the maximum overpressure of the sonic boom by 25.1%, and decreased the root bending moment by 20.3%. Overall, MG-MOGBA can efficiently solve high-dimensional multi-objective aerodynamic optimization problems and has good scalability for the dimensions of design objectives and variables (Figure 1).

References

[1] P. Champasak, N. Panagant, N. Pholdee, S. Bureerat, A. R. Yildiz, Self-adaptive many-objective meta-heuristic based on decomposition for many-objective conceptual design of a fixed wing unmanned aerial vehicle, Aerospace Science and Technology 100 (2020) 105783.

[2] Y. Tian, X. Zhang, R. Cheng, C. He, Y. Jin, Guiding evolutionary multi-objective optimization with generic front modeling, IEEE transactions on cybernetics 50 (2018) 1106– 1119.

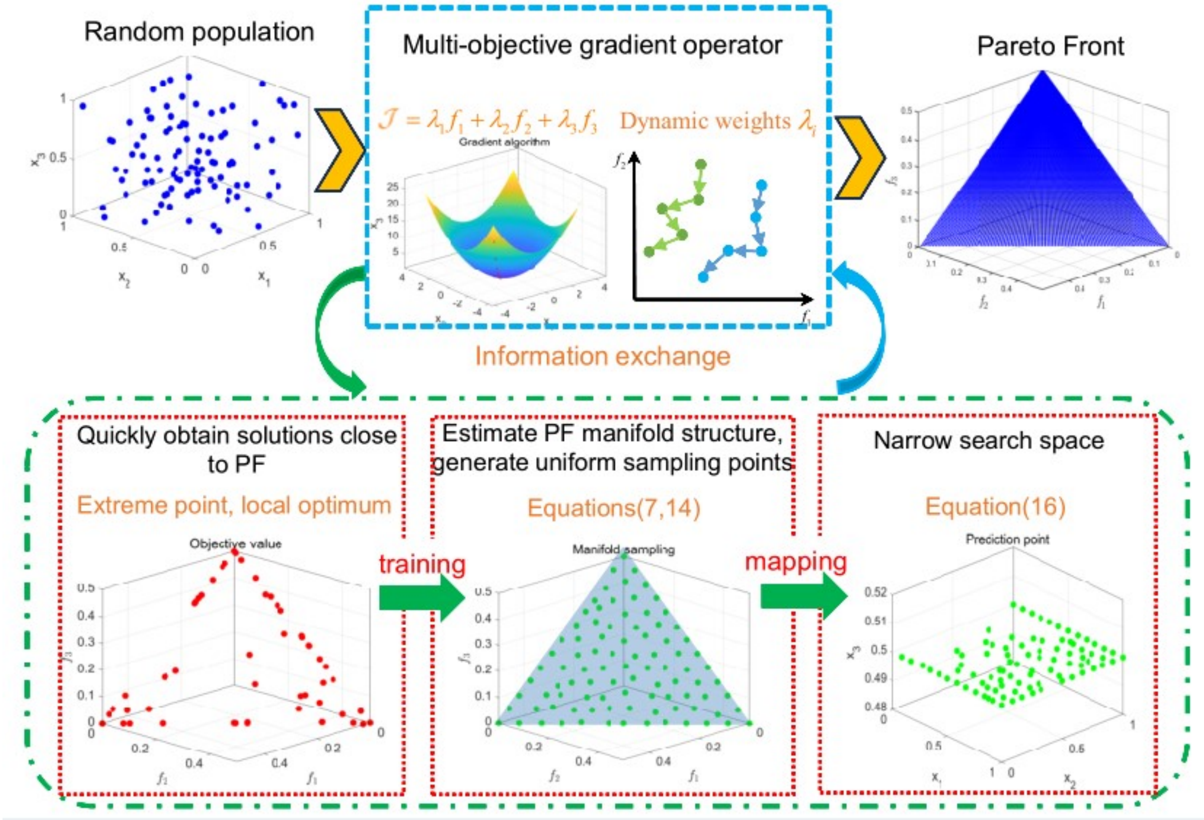


Figure 1: *Schematic diagram of the MG-MOGBA*

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 248-3

Unsteady Numerical Simulation of Icing Process on Oscillating Airfoils

Chenglin Tian[1], Wei Guo[1], Ning Zhao[1] and Chunling Zhu[1]*

[1] College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, No. 29, Yudao Street, Qinhuai District, Nanjing City, Jiangsu Province, China
clzhu@nuaa.edu.cn (Chunling Zhu)

Key Words: *Aircraft icing, oscillating airfoil, unsteady flow, unstructured overlapping grid, Eulerian two-phase flow, icing thermodynamics*

Icing on rotor blades can reduce aerodynamic characteristics and pose a serious threat to helicopter flight safety. The rotor undergoes periodic movement with variable pitch during forward flight, which can be simplified as the pitching oscillation process of the rotor airfoil. The unsteady characteristic of icing on oscillating airfoils presents a challenging problem compared to icing on fixed wings.

To address this issue, numerical simulations of the unsteady airflow field in an unstructured overlapping grid system and the impact characteristics of supercooled water droplets based on the Euler method were conducted. Subsequently, the unsteady icing calculation method was improved based on Myers icing model. Additionally, a dynamic grid generation method called DGRBF (Radial Basis Function based on Delaunay Graph) was developed for multi-step icing calculations. A set of unsteady numerical simulation methods has been developed for icing on oscillating airfoils.

The study analysed the effects of parameters such as average angle of attack, oscillation amplitude, and oscillation frequency on the icing process. In comparison to the static condition, the pitching motion of the airfoil takes into account the periodic variation of the angle of attack. The presence of a larger range of supercooled water droplets on the airfoil leads to increased icing and a higher icing limit. The final ice shape is not significantly affected by the oscillation frequency and amplitude of the airfoil's pitching motion at the average angle of attack. Additionally, there is no significant difference between the ice shape under the pitching motion and the ice shape at rest.

Furthermore, the comparison between the numerical simulation results and the icing wind tunnel test results revealed a strong correlation in the macroscopic characteristics, including the icing limit, ice angle position, and maximum icing thickness. This confirms the accuracy and reliability of the calculation method. As a result, the developed unsteady icing numerical simulation method exhibits great potential for engineering applications.

References

[1] M. Morelli, B. Zhou and A. Guardone. Acoustic characterization of glaze and rime ice structures on an oscillating airfoil via fully unsteady simulations. Journal of the American Helicopter Society, 65(4):1–12, 2020.

[2] S. Min and K. Yee. Numerical investigation of the unsteady effect owing to oscillation on airfoil icing. International Journal of Heat and Mass Transfer, 203:123791, 2023.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 248-4

Parameter Reduction for Adjoint Based Aerodynamic Shape Optimisation of Gas Turbine Tips

Guoxi Ye and Ning Qin*

Department of Mechanical Engineering, University of Sheffield, Sheffield, S1 3JD, South Yorkshire, England, United Kingdom, n.qin@sheffield.ac.uk

Key Words: *Aerodynamic shape optimisation, Adjoint method, tip leakage flow*

Blade tips play an important role in the efficiency of gas turbines in minimising the adverse effects of the tip leakage flows. Various tip shapes such as squealers and winglets have been proposed and investigated for improving the performance. The advance of adjoint based methods have allowed for more flexible tip shape to be parameterized with hundreds or thousands design variables. However, the flexibility also increases substantially the dimension of the design space so that the searching for the optimum becomes a lot more costly. In this paper, we present a parameter reduction method based on the sensitivity obtained from the adjoint solution so that the optimiser can narrow down on the key design parameters for efficient shape optimisation without the constraints of pre-specified topologies.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 248-5

Euler-Lagrangian Method of SLD Droplet Collection Efficiency Calculation Based on Particle Parallelism

WenLong Xie[1], Qingyong Bian[1] and Chunling Zhu[1]*

[1] College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics - NUAA, No.29, Yudao Street, Qinhuai District, Nanjing City, Jiangsu Province, China
clzhu@nuaa.edu.cn (Chunling Zhu)

Key Words: *Euler-Lagrange, SLD, droplet collection efficiency, particle parallelism*

The calculation of water droplet impact characteristics is essential for predicting aircraft icing. The Lagrangian method is commonly used because it can track particle trajectories, deformation, breakup, and splashing. However, its 3D computational speed has been criticized. This paper proposes a parallel strategy based on particle parallelism, which enhances the computational efficiency of the Lagrangian method. The method partitions the domain by particles to calculate the water drop field, as shown in Figure 1, maximizing parallelism efficiency. This approach's effectiveness has been demonstrated on two-dimensional NACA23012 airfoils, with a remarkable acceleration effect. As shown in Figure 2, the time required for the case is reduced to one-fourth of the time required for a single core with an increase in cores. It is expected that extending this method to 3D and achieving parallelism on a larger scale will further reduce 3D Lagrangian computation times.

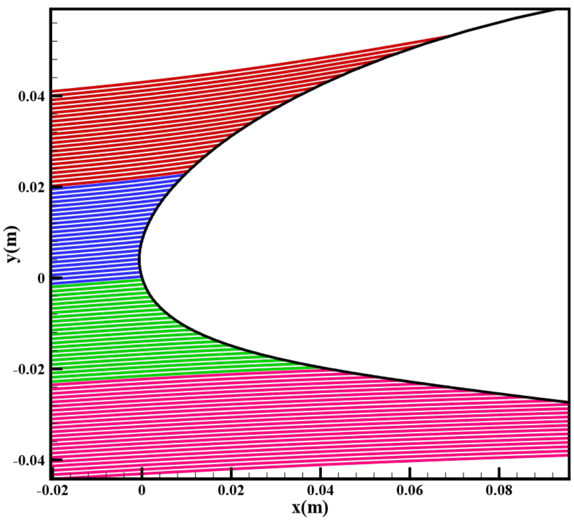


Figure 1 4-core system droplet trajectory

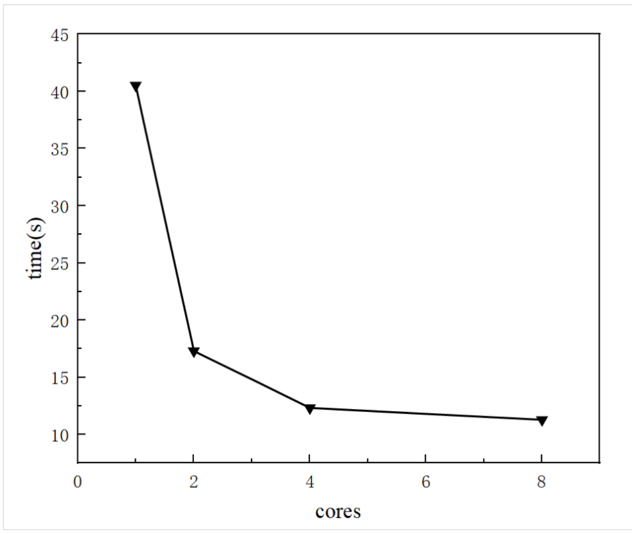


Figure 2 Cost of time for different cores

References

[1] Bellosta, T., Baldan, G., Sirianni, G., & Guardone, A, Lagrangian and Eulerian algorithms for water droplets in in-flight ice accretion. Journal of Computational and Applied Mathematics, 429,2023

[2] Baldan, G., Bellosta, T, & Guardone, A, Efficient Lagrangian particle tracking algorithms for distributed-memory architectures. Computers & Fluids, 256,2023

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 267

The Impact of Multi-Disciplinary Optimization, Artificial Intelligence and Uncertainty Quantification for a Greener Aviation and Transport System

Chair: Alberto Clarich

ESTECO SpA, AREA Science Park, Padriciano 99, 34149 Trieste, Italy, www.esteco.com
clarich@esteco.com

Session Abstract

Key Words: *Multi-Disciplinary Optimization, Artificial Intelligence and Uncertainty Quantification, Greener Aviation, Transport System*

Industrial design is facing new challenges in a world threaten by global warming, economical and geo-political instability. In this contest, the need of decreasing emissions in air, for all the activities related to aviation, transport and renewable energy, has become an utmost priority, also considering the commitments made by the European Commission for the next years.

In particular, net greenhouse gas emissions will have to be reduced by at least 55% by 2030, and at the same time it will be necessary to fulfil the growing energy demand of all the EU industrial countries.

Following this perspective, the new emerging technologies in Industrial design, many of them already developed in the framework of several European Programme of Research & Innovation projects, need to reach soon a TRL of over 8-9 level, which means that they need to be implemented in a complete and qualified system, ready for the commercialization.

This STS addresses all the emerging technologies which aim to reach a greener transition of the industrial design, and includes in a non-exhaustive way efficient Multi-Disciplinary Optimization (MDO) methodologies, Design based on Artificial Intelligence, whether by Surrogate Models, Machine Learning, Reduced Order Models or Multi-Fidelity Models, and Robust Design or Design under Uncertainties.

This STS invites experts from industry, research institutions and universities, to present their recent studies and applications of the new emerging technologies above mentioned in the design of greener, more efficient and more sustainable aviation and transport systems, or renewable energy.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 267-1

Regularized Reduced Order Models for Convection-Dominated Flow Simulation: Stabilization, Prediction, Sensitivity and Control

Maria Strazzullo[1]*

[1] Politecnico di Torino, Department of Mathematical Sciences "Giuseppe Luigi Lagrange", Corso Duca degli Abruzzi, 24, 10129, Torino, Italy,
maria.strazzullo@polito.it

Key Words: Numerical Stabilization, Reduced Order Models, Convection-Dominated Flows, Navier-Stokes Equations

This presentation focuses on the development of tailored reduced-order models (ROMs) designed for bifurcating nonlinear parametric partial differential equations (PDEs). In this framework, multiple solutions can emerge for a given physical and/or geometrical parametric instance. Traditional certified reduced basis techniques struggle to accurately represent bifurcating phenomena, as error estimations become unreliable when bifurcations occur. Consequently, ROMs for bifurcating nonlinear PDEs have primarily relied on data compression strategies, such as proper orthogonal decomposition.

The primary objective of the talk is to propose an innovative tailored greedy algorithm for bifurcating nonlinear PDEs based on deflation-based strategies. These algorithms can (i) simultaneously certify multiple behaviors of the system through the deflated-greedy approach and (ii) identify the parameter responsible for the non-uniqueness of the solution through an adaptive greedy approach, even when provided with limited information about the parametric space.

The deflated-greedy method leverages various techniques, including deflation and continuation, to enrich the reduced space with bifurcating solutions. On the other hand, the adaptive-greedy approach exploits the non-differentiability of the solution with respect to the bifurcating parameter.

The effectiveness of these strategies is tested on the Navier-Stokes equations in a sudden expansion channel, featuring three coexisting solutions for a single parameter. The results are compared in terms of accuracy and error certification against the high-fidelity solution, contrasting with standard greedy and proper orthogonal decomposition methods.

This talk show results from works with several collaborators: F. Ballarin, C. Canuto, T. Chacón Rebollo, M. Girfoglio, A. Ivagnes, T. Iliescu, G. Rozza.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 267-2

Bayesian Multidisciplinary Optimization of Transonic Strut-Braced Wing Aircraft Based on Multi-Fidelity Reduced Order Surrogate Model

Shon Mori[1,*], Mathieu Balesdent[1], Loic Brevault[1], Sylvain Dubreuil[2], Cédric Liauzun[3]

[1] ONERA - DTIS, Université Paris Saclay, 91123 Palaiseau, France, www.onera.fr/en
shon.mori@onera.fr, mathieu.balesdent@onera.fr, loic.brevault@onera.fr
[2] ONERA - DTIS, Université de Toulouse, 31055 Toulouse, France, sylvain.dubreuil@onera.fr
[3] ONERA - DAAA, Université Paris Saclay, 92320 Châtillon, France, cedric.liauzun@onera.fr

Key Words: *Multi-fidelity surrogate modeling, Gaussian process, reduced order model, proper orthogonal decomposition, multidisciplinary analysis and optimization, strut-braced wing*

Research Context

The civil aviation industry grapples with the challenge of minimizing environmental impact. Innovative concepts, like the High Aspect Ratio Strut- Braced Wing (HARSBW), offer potential efficiency gains [2], but their Multi-Disciplinary Analysis and Optimization (MDAO) face challenges due to transonic flight and complex aerodynamic-structural coupling [3]. Numerical optimization of such a concept requires the use of high-fidelity solvers, and the induced numerical cost limits its practical application. A Bayesian multi-fidelity optimization framework is investigated in this paper as an alternative to enable the parametric study of this concept.

Prior Art

HARSBW's appeal lies in its aerodynamic efficiency, achieved through a higher Aspect Ratio (AR) strut-braced wing [3]. Existing studies employ MDAO, addressing aero-structural coupling complexities.

To assess preliminary aircraft design performance in MDAO, Fluid-Structure Interaction (FSI) analysis involving aero-structural Multidisciplinary Analysis (MDA) must be conducted for the parameterized design domain. Traditional dense sampling of the design space for global optimization is computationally expensive and intractable. This study employs a novel approach of using Proper Orthogonal Decomposition (POD) for dimension reduction of pressure and displacement field coupling variables and constructing Gaussian Process (GP) surrogate models for MDA coupling [1]. Multi-fidelity models utilize low and high-fidelity aerodynamic solvers, such as Vortex Lattice Method (VLM) and CFD Euler, achieving accuracy and efficiency.

The study aims to develop a multi-fidelity Bayesian MDAO by refining the multi-fidelity POD-GP MDA process and exploring enrichment strategies to efficiently obtain the global optimum in the preliminary design of HARSBW.

Problem Statement and Expected Outcomes

This paper investigates enrichment strategies for the multi-fidelity reduced-order surrogate model (MF-POD+GP) in MDAO for preliminary aircraft design. The study emphasizes an active learning process leveraging multi-fidelity GP's uncertainty quantification for efficient model enrichment by maximizing the accuracy of the model in the optimal regions of interest and minimizing computational costs. Enrichment targets both the POD basis and GP surrogate model, capturing the full range of physical phenomena in the design domain. The expected outcome is a complete and efficient MDAO process on the refined MF-POD+GP model with active learning applied to the HARSBW preliminary design.

References

[1] Gaspard Berthelin, Sylvain Dubreuil, Michel Salaün, Nathalie Bartoli, and Christian Gogu. Disciplinary proper orthogonal decomposition and interpolation for the resolution of parameterized multidisciplinary analysis. International Journal for Numerical Methods in Engineering, 123(15):3594–3626, 2022.

[2] Frank H Gern, Andy Ko, Erwin Sulaeman, John F Gundlach, Rakesh K Kapania, and Raphael T Haftka. Multidisciplinary design optimization of a transonic commercial transport with strut-braced wing. Journal of Aircraft, 38(6), November 2001.

[3] Ohad Gur, Joseph A Schetz, and William H Mason. Aerodynamic considerations in the design of truss-braced-wing aircraft. Journal of Aircraft, 48(3):919–939, 2011.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 267-3

Uncertainty Quantification Method Based on Reduced Order Models for Aeronautics

Luca Battaglia[1,*], Alberto Clarich[2] and Rosario Russo[3]

ESTECO SpA, AREA Science Park, Padriciano 99, 34149 Trieste, Italy, www.esteco.com
[1] battaglia@esteco.com, [2] clarich@esteco.com, [3] russo@esteco.com

Key Words: *Uncertainty Quantification, Reduced order model (ROM), Aeronautical flow*

Many industrial processes and products are affected by uncertainties arising from several factors, such as dimensional tolerances, manufacturing errors, fluctuating or unknown operational parameters. In engineering design, the uncertainties of the design parameters are transferred to the system responses, in such a way that they must be described in terms of statistical distributions rather than single deterministic values.

To deal with problems affected by a large number of uncertainties and expensive simulation time, e.g. CFD analyses, it is particularly important to develop methodologies which are at the same time accurate and that can rely on a limited number of sample evaluations.

In this paper, we propose an Uncertainty Quantification (UQ) method based on Reduced Order Model (ROM) and non-intrusive Polynomial Chaos Expansion (PCE) to efficiently compute the uncertainty propagation of a vectorial field of interest.

The method is first applied to a RANS flow over an airfoil with uncertain angle of attack and Mach number, to evaluate its accuracy in terms of results and efficiency in terms of limited number of CFD samples evaluation, compared to other methods.

The method is then applied to the Euler flow over a supersonic jet aircraft, to estimate average and standard deviation of the pressure field in each grid point, due to the propagation of uncertain operational parameters.

References

[1] J. Weiss. A Tutorial on the Proper Orthogonal Decomposition. doi:10.2514/6.2019-3333. URL https://arc.aiaa.org/doi/abs/10.2514/6.2019-3333.

[2] G. Blatman and B. Sudret. Adaptive sparse polynomial chaos expansion based on least angle regression. Journal of Computational Physics, 230(6):2345–2367, 2011. ISSN 0021- 9991. doi: https://doi.org/10.1016/j.jcp.2010.12.021.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 267-4

Modeling of 3D Printed Soft Pneumatic Actuators

Lorenzo Torzini[1], Lapo Governi[2], Luca Puggelli[3] and Francesco Buonamici[4]*

Department of Industrial Engineering, University of Florence,
Via di S. Marta 3, 50139 Florence, Italy, www.dief.unifi.it/
[1] lorenzo.torzini@unifi.it, [2] lapo.governi@unifi.it, [3] luca.puggelli@unifi.it,
[4] francesco.buonamici@unifi.it

Key Words: *Pneumatic actuators, 3D printing, soft actuators, Finite element method*

Soft Pneumatic Actuators (SPAs) are characterized by a rather simple structure with a series of inner chambers that are pressurized to trigger a deformation and produce forces. In this context, Additive Manufacturing (AM) was successfully applied to produce enhanced SPAs by exploiting design freedom and customizing them for specific tasks. A faithful prediction of the behavior of a SPA would allot the production of optimal SPAs avoiding a cumbersome trial-and-error approach. Nevertheless, performing FEM analyses of 3D printed soft structures it's not a trivial task. This study aims at the development of a FEM model to simulate a generic 3D printed bellow SPA performing a pure planar bending motion under pressurization that can be used with different SPA design parameters.

The obtained model is used to evaluate properties like the bending angle and exerted forces given a certain combination of design parameters. The performances of a bellow SPA are influenced by several factors (main dimensions, material, operating pressure, shape of the chambers, etc.); this study has considered three main contributors to the bending behavior: wall chambers thickness, number of bellow segments and operating pressure. Many configurations were printed in TPU using FFF printer and tested using a physical rig that measures the bending angle of the actuator and the generated forces.

FEM analyses were carried out within Ansys 2019 using a static structural analysis with the actuator fixed at one extremity and pressurized internal surfaces. As hyper-elastic material, Mooney-Rivlin 5-parameters model was used, according to [1]. Material parameters were identified for a limited number of configurations, due to high computational costs, through an optimization process that minimized the bending angle error with respect to actual values. Other actuator configurations were simulated using various sets of parameters derived from the optimization stage and confronted with the physical test results. The results show a valid approximation obtained across the different configurations of the actuators; insights on plausible material parameters for generic 3D printed bellow SPAs were identified. In conclusion, the study has allowed the development of an effective methodology to simulate the behaviour of a generic 3D printed TPU bellow SPA with satisfying level of approximation.

References

[1] Tawk, Charbel and Alici, Gursel, Finite Element Modeling in the Design Process of 3D Printed Pneumatic Soft Actuators and Sensors. Robotics, 2020.

STS 267-5

Fundamental Thermoelastic Behavior Modeling for L-PBF Additive Manufacturing

Juha Jeronen[1]*, Tero Tuovinen[2] and Matti Kurki[3]

JAMK University of Applied Sciences, 40101 Jyv¨askyl¨a, Finland, www.jamk.fi/en,
[1] juha.jeronen@jamk.fi, [2] tero.tuovinen@jamk.fi, [3] matti.kurki@jamk.fi

Key Words: *Additive Manufacturing, Laser Powder Bed Fusion (L-PBF), Selective Laser Melting (SLM), Mathematical Modeling, Coupled Problem, Multiphysics, Thermomechanical Analysis, Viscoelasticity, Axially Moving Materials*

Additive manufacturing (3D printing) of metals has risen in popularity in industrial applications. A key aspect is that there are very few or no geometric restrictions. Particularly, the 3D printing of metals allows one to make complex, durable, custom parts. [1]

For the last few years, our group has investigated [2] the thermomechanical behavior of 3D printing of metals in the laser-based powder bed fusion (L-PBF) process, also known as selective laser melting (SLM). Here we highlight the main results of the research.

We construct a thermoviscoelastic continuum model for the case where a thin fin is being printed at a constant velocity. We use a coordinate frame that moves with the printing laser, and apply a Eulerian perspective to the moving solid. We consider a steady state similar to those used in the analysis of production processes in the process industry, in the field of research known as axially moving materials. We demonstrate the model with material parameters for 316L steel.

As our main result, we obtain the steady-state deformation shape in the wake of the focus spot of the laser in a two-dimensional setting. Another important result are nondimensional parameters derived from a one-dimensional model. Similarly to how the Reynolds number governs flows, these parameters govern the behavior of the printing process.

Finally, we extend traditional linear rheology by introducing a kinetic inductor element, inspired by electrical RLC circuits, to complement the classical spring and dashpot elements. This adds the inertial response of the material to the constitutive law. This modification becomes important in settings with quickly varying loading. Although the effect is negligible in most cases for steel, for soft materials under acoustic or ultrasound loading, the magnitude of the inertial term becomes important.

References

[1] John O. Milewski. Additive Manufacturing of Metals. Springer, 2017, Springer Series in Materials Science, vol. 258. doi.org/10.1007/978-3-319-58205-4

[2] Juha Jeronen, Tero Tuovinen, and Matti Kurki. Fundamental Mathematical Modeling of Additive Manufacturing. Springer, 2024, Springer Tracts in Additive Manufacturing. doi.org/10.1007/978-3-031-47250-3

STS 268 A & B

Stability and Sensitivity Methods for Flow Control and Industrial Design

Chair: Eusebio Valero

Universidad Politécnica de Madrid (UPM), Plaza Cardenal Cisneros 3, 28040 Madrid, Spain
eusebio.valero@upm.es, https://www.upm.es

Session Abstract

Key Words: *Stability, Sensitivity methods, Flow control, Industrial design*

Within the framework of the European Project SSECOID: Stability and Sensitivity Methods for Flow Control and Industrial Design, this Special Technology Sessions (STS) aims to uncover ongoing advances in numerical and computational methods for fluid mechanics. This broad goal encloses not only the development of standard numerical schemes for the integration of the Navier Stokes equations, but also state-of-the-art Lattice Boltzmann or the difficulties and progress in the application of high-order schemes to more realistic industrial configurations, in particular the implementation of prevailing methods in present and future computational platforms, with an eye on exascale computing.

Moreover, the huge amount of data generated by numerical tools needs further postprocessing: data assimilation methods and machine learning as a way to obtain the flow sensitivity under perturbation, eventually linked to stability, optimization and control, can provide very valuable information about the flow. Several algorithms, such as DMD, POD, SPOD or Resolvent can obtain very important information that helps to identify relevant features, which are critical to understand the flow behaviour and, finally, will provide valuable information to control it.

New developments and application of those methodologies to feature detection, optimization strategies and control are welcome.

Contributing papers to the session parts A and B:
Part A: STS 268-1 to STS 268-4; **Part B:** STS 268-5 to STS 268-8.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 268-1

Global Stability Analysis of Wavy-Surface Induced Laminar Separation Bubbles

Mohammad Moniripiri[1,*], Daniel Rodriguez[2] and Ardeshir Hanifi[1]

[1] FLOW, Department of Engineering Mechanics, KTH Royal Institute of Technology, 10044 Stockholm, Sweden. momp@kth.se, hanifi@kth.se

[2] Universidad Politécnica de Madrid, ETSIAE-UPM, Plaza del Cardenal Cisneros 3, 28040 Madrid, Spain, daniel.rodriguez@upm.es

Key Words: *Global stability analysis, Laminar separation bubble (LSB), Surface waviness.*

Laminar Separation Bubbles (LSBs) may adversely affect aerodynamic performance by causing laminar-turbulent transition. The stability and dynamics of flat plate pressure-gradient induced LSBs have been studied extensively in the literature [1]; however, surface-induced LSBs, especially those arising from wavy surfaces, have received less attention [2]. Surface waviness, by imposing successive adverse and favourable pressure gradient regions on the wall, creates a complex flow regime, and even can distort the streamlines of flow in a way that affects the stability characteristics of LSBs and triggers three dimensional instabilities different to the ones present in flat plate pressure-gradient-induced LSBs.

In this study, the onset of flow separation for numerous sinusoidal surface waviness at various Reynolds numbers is identified. A scaling law is then established, enabling the prediction of the geometric parameters leading to flow separation at a specific Reynolds number. Then, several different cases are selected for further two- and three- dimensional stability analysis using linear and non-linear frameworks.

Regarding two-dimensional global linear analysis, an extensive study on effects of domain size and boundary conditions is carried out, and it is shown that choosing the proper domain size and boundary conditions is essential to recover the growth rate of two-dimensional planar waves accurately.

To investigate the instabilities of three-dimensional disturbances in a spanwise homogenous baseflow, a two-dimensional global stability approach is utilized, and the resulting large eigenvalue problem is solved using parallel sparse linear algebra techniques. It is shown that, similar to pressure-gradient-induced LSBs on flat plates, three-dimensional instabilities are triggered prior to two dimensional ones. Interestingly, as the curvature of surface waviness increases, new zero-frequency three-dimensional unstable modes localized around the rear part of LSB, not observed in flat plate pressure-gradient-induced LSBs, with relatively large exponential growth rates become the dominant primary instability. Moreover, it is observed that as the curvature increases, the zero-frequency unstable mode that exists for flat-plate pressure-gradient-induced separation bubbles becomes stable and the new localized modes become unstable. Further, three-dimensional DNS will be carried out to identify the transition mechanism for different cases with different curvatures.

References

[1] Rodríguez, D., Gennaro, E.M. and Souza, L.F., 2021. Self-excited primary and secondary instability of laminar separation bubbles. *Journal of Fluid Mechanics*, *906*, p.A13.

[2] Ehrenstein, U. and Gallaire, F., 2008. Two-dimensional global low-frequency oscillations in a separating boundary-layer flow. *Journal of Fluid Mechanics*, *614*, pp.315-327.

STS 268-2

Active Control of Separated Flow over a Bump under Unsteady and Turbulent Conditions

Himpu Marbona[1,2]*, Daniel Rodriguez[1], Alejandro Martínez-Cava[1,3] and Eusebio Valero[1,2]

[1] Universidad Politécnica de Madrid, Plaza Cardenal Cisneros 3, 28040 Madrid, Spain
himpu.marbona@alumnos.upm.es, daniel.rodriguez@upm.es, alejandro.martinezcava@upm.es, eusebio.valero@upm.es

[2] Center for Computational Simulation, UPM, Campus de Montegancedo, 28660 Madrid

[3] Instituto Universitario "Ignacio Da Riva", UPM, Plaza Cardenal Cisneros, 3, 28040, Madrid, Spain

Keywords: *Separated flow, laminar-turbulent transition, low-pressure turbines*

Low-pressure turbines (LPT) encounter laminar and/or turbulent boundary layer separation during operation in a low-Reynolds regime [1,2]. This phenomenon is frequently analyzed using a wall-bounded bump geometry, which simulates the suction side of LPT blades. Separated flow leads to adverse effects on aerodynamics and performance, prompting the exploration of flow control strategies aimed at reducing the size of the separated flow region to enhance efficiency [3].

This study investigates laminar and turbulent separation over a wall-bounded bump geometry. Direct Numerical Simulations (DNS) results are examined to unveil the impact of harmonic inflow and flow actuations with zero-net-mass-flow. Three distinct scenarios involving KH-instability and/or vortex clusters are revealed. Furthermore, turbulent boundary layer separation is explored using wall-modeled large-eddy simulations (WMLES) to comprehend the influence of pressure gradient and freestream turbulence on the separation region with aim for active control strategies.

Acknowledgments

This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sklodowska Curie grant agreements No 955923-SSECOID and 101019137-FLOWCID. D.R. also acknowledges funding by the Government of the Community of Madrid within the multi-annual agreement with Universidad Politécnica de Madrid through the Program of Excellence in Faculty (V-PRICIT line 3).

References

[1] R. E. Mayle, The Role of Laminar-Turbulent Transition in Gas Turbine Engines (1991)

[2] D. Coull and H. P. Hodson. Unsteady boundary-layer transition in low-pressure turbines. J. Fuid Mech., 681:370–410, 2011.

[3] R. J. Volino, O. Kartuzova, and M. B. Ibrahim. Separation Control on a Very High Lift Low Pressure Turbine Airfoil Using Pulsed Vortex Generator Jets. J. of Turbomachinery, 133(4), 04 2011.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 268-3

Influence of Free-stream Turbulence and Discrete Roughness Elements on the Receptivity of Two-dimensional Boundary Layers

Emilio López Figueiras[1,2]*, Daniel Rodríguez[2] and Jean-Baptiste Gouriet[1,3]

[1] Aeronautics and Aerospace Department, von Karman Institute for Fluid Dynamics,
Waterloosesteenweg 72,1640 Sint-Genesius-Rode, Belgium,
[1] emilio.lopez.figueiras@vki.ac.be, [3] jean-baptiste.gouriet@vki.ac.be

[2] Applied Mathematics in Aerospace Engineering Department, ETSI Aeronaútica y del Espacio,
Plaza Cardenal Cisneros 3, 28040, Madrid (Spain); daniel.rodriguez@upm.es

Keywords: *Receptivity, Transition to Turbulence, Discrete Roughness Element, Freestream Turbulence, Direct Numerical Simulation, Flat Plate*

Among the strategies that could improve the efficiency of nowadays aircraft to decrease their carbon footprint, the reduction of aerodynamic drag is one of the most important. In that sense, friction drag is the main contributor to the total aerodynamic drag, and the delay or control of the processes that lead to the flow transition to turbulence plays a key role in the increase of aerodynamic efficiency. Receptivity is the mechanism that defines the way that disturbances penetrate into the boundary layer and have its origin in the incoming free-stream perturbations, in the surface roughness of the aerodynamic devices or in the interaction between them.

This work will treat the numerical receptivity analysis of incompressible, two-dimensional boundary layers to free-stream turbulence, discrete roughness elements and their interaction; giving more knowledge about the physical mechanisms involved in the transition process and helping to the development of strategies to its delay and control, which can reduce the aerodynamic drag produced by turbulence.

During several decades, the receptivity of two-dimensional boundary layers has been an intensive field of research. In this kind of flows, Tollmien-Schlichting waves are the main sources of instabilities and might be produced by perturbations like discrete roughness elements or free-stream turbulence, as it is described in [1].

In this work, the influence of the leading edge will be included in the analysis, using the well-known ”modified super-ellipse”, which is defined in the receptivity analysis to free-stream vorticity modes given by [2]. The domain shape, size and base flow will be common with [2] too. Additionally, isotropic free-stream turbulence will be generated at the far-field boundaries, following the work by [3] in a similar way to the one applied to 2

a three-dimensional boundary layer case in [4]. The receptivity to a discrete roughness element will be also an object of study, taking the experimental work from [5] as a reference to establish its position along the flat plate chord (based on the roughness Reynolds number Re_{kk}) or its height (compared with the boundary layer displacement thickness at that location), as well as the boundary layer development along the flat plate geometry. Taking an experimental work as a reference will be useful to validate the numerical results obtained in this work. The planned simulations will cover cases that will vary the position of the roughness element along the flat plate chord with and without the presence of freestream turbulence, responding to the need stated in [6] to study parameters like the turbulence intensity, which is independent from the already studied roughness Reynolds number or the roughness element height.

All the simulations will be run with the spectral elements DNS code for incompressible flows *Nek5000*, using the computational resources already allocated at the Tier-1 Computing service delivered by the Flemish Supercomputer Center (VSC). The simulations and the analysis of the results are expected to be completed before the second half of February 2024.

References

[1] W. S. Saric, H. L. Reed and E. J. Kerschen, Boundary layer receptivity to free-stream disturbances, *Annual Review of Fluid Mechanics*, Vol. **34**, pp. 291–319, 2002.

[2] L.-U. Schrader, L. Brandt, C. Mavriplis and D. S. Henningson, Receptivity to freestream vorticity of flow past a flat plate with elliptic leading edge, *Journal of Fluid Mechanics*, Vol. **653**, pp. 245–271, 2010.

[3] P. Schlatter, *Direct numerical simulation of laminar-turbulent transition in boundary layer subject to free-stream turbulence*, Master's thesis, Diploma Thesis Winter Term 2000/2001, Royal Institute of Technology. Department of Mechanics, 2001.

[4] L. de Vincentiis, D. S. Henningson and A. Hanifi, Transition in a infinite swept-wing boundary layer subject to surface roughness and free-stream turbulence, *Journal of Fluid Mechanics*, Vol. **931**, pp. A24, 2022.

[5] I. B. de Paula, W. Wu¨rz, M. T. Mendon¸ca and M. A. F. Medeiros, Interaction of instability waves and a three-dimensional roughness element in a boundary layer, *Journal of Fluid Mechanics*, Vol. **824**, pp. 624–660, 2017.

[6] M. A. Bucci, S. Cherubini, J.-Ch. Loiseau and J–Ch. Robinet, Influence of freestream turbulence on the flow over a wall roughness *Phys. Rev. Fluids*, Vol. **6**, pp. 063903, 2021.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 268-4

Numerical Studies on the Influence of Steps on Secondary Instability Mechanisms in Crossflow-Dominated Boundary Layers

Biagio Ambrosino[1,2]*, Francesco Tocci[1], Alexander Theiß[1] and Stefan Hein[1]

[1] German Aerospace Center - DLR, Institute of Aerodynamics and Flow Technology, 37073 Göttingen, Germany; biagio.ambosino@dlr.de, francesco.tocci@dlr.de, alexander.theiss@dlr.de, stefan.hein@dlr.de

[2] Universidad Politécnica de Madrid, Plaza Cardenal Cisneros 3, 28040 Madrid, Spain

Keywords: *Stability analysis, crossflow, swept wing, surface irregularities, fluid dynamics*

Surface irregularities such as steps, roughness and waviness can have a significant impact on the laminar-to-turbulent transition process over wings and tails of commercial aircraft. In this work, the effect of forward-facing steps (FFS) on subsonic crossflow-vortex dominated boundary layers is investigated. The geometry under study is a swept-wing, which reproduces the experimental setup of Rius-Vidales et al. [1] in the Low Turbulence Tunnel (LTT) at the Delft University of Technology. The setup is characterized by the presence of a forward-facing step with different heights. Results from direct numerical simulations (DNS), conducted in previous work on this geometry, are here employed as the base flow for the instability analysis. In particular, two-dimensional linear stability theory (LST-2D) and three-dimensional parabolized stability equations (PSE-3D) are employed to study the influence of two step heights on the secondary instabilities of crossflow vortices and identify how those perturbations change compared to the case without step. Furthermore, the analysis aims to assess the instability characteristics of the flow in the region downstream of the step and to identify the unstable disturbances that are developing. In order to cope with a base flow that is periodic in the direction parallel to the leading edge and features a slow variation along the crossflow-vortex axis, the stability problem is formulated in a non-orthogonal coordinate system, following the approach proposed by Li *et al.* [2] and later employed by Groot *et al.* [3] to perform LST-2D computations on an experimentally measured crossflow-vortex-dominated flow over a forward-facing step.

Acknowledgments

This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sk lodowska-Curie grant agreement No. 955923.

References

[1] Rius-Vidales, A.F. and Kotsonis, M., *Impact of a forward-facing step on the development of crossflow instability*, Journal of Fluid Mechanics, Vol. 924, A34.

[2] Li, Fei and Choudhari, Meelan M., *Spatially Developing Secondary Instabilities in Compressible Swept Airfoil Boundary Layers*. Theoretical and Computational Fluid Dynamics, Vol. 25, 2011, pp. 65-84.

[3] Groot, K. and Eppink, J., *Stability Analysis of the Flow over a Swept Forward-Facing Step using PIV Base Flows in a Nonorthogonal Coordinate System*. AIAA 2021-2908 Aviation Forum, 2021.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3-–7 June 2024, Lisbon, Portugal

STS 268-5

Competition between Stall Cells and Low-Frequency Global Modes in Transitional Flows around Airfoils Near Stall

Konstantinos Sarras[1,*] **and Olivier Marquet** [2]

ONERA – The French Aerospace Lab, Université Paris Saclay, F-92190 Meudon, France,
[1] konstantinos.sarras@onera.fr, [2] olivier.marquet@onera.fr, www.onera.fr/en

Keywords: *Linear stability, RANS modelling, airfoil stall*

Turbulent flows around airfoils near stalling conditions may be characterized by low frequency oscillations during which the flow alternates between attached and detached states, and by stall cells, that are spanwise modulation of the flow on the suction side of the airfoil. Recent studies [1, 2] have shown that the onset of these two phenomena can be captured using global stability analysis of the fully turbulent mean-flow estimated within the framework of the Reynolds-Averaged Navier Stokes (RANS) equations. In this talk, we will investigate the competition between these two global instabilities but for a transitional flow over a NACA 0012 airfoil at the Reynolds number $Re = 90000$.

The transitional nature of the flow is modelled using the linear eddy-viscosity model developed by Spalart & Allmaras that is coupled here with a correlation-based algebraic transition model. Similarly to the fully turbulent cases mentionned before, the branch of steady solutions exhibits a *S* shape curve with two saddle-node bifurcations. The solutions of the high-lift branch are however characterized by the existence of a laminar separation bubble (LSB) at the leading edge. The global stability analysis of these steady solutions is based on the full linearization of the governing discrete equations, including the turbulence model, the transition model and the numerical stabilization terms. It reveals the existence of two unstable modes: a two-dimensional low-frequency mode similar to that found by [2] and a three-dimensional zero-frequency mode similar to that found by [1]. The competition between these two modes is investigated by following the corresponding eigenvalues along the branch of steady solutions. Thus, we have identified the critical angles for each mode and shown that the three-dimensional modes become unstable prior to the two-dimensional ones, for that particular case.

References

[1] F. Plante, J. Dandois, S. Bennedine, E. Laurendeau, D. Sipp´ *Link between subsonic stall and transonic buffet on swept and unswept wings: from global stability analysis to nonlinear dynamics*. Journal of Fluid Mechanics, 908, A16, 2021.

[2] D. Busquet, O. Marquet, F. Richez, M. Juniper, D. Sipp *Bifurcation scenario for a two-dimensional static airfoil exhibiting trailing edge stall*. Journal of Fluid Mechanics, 928, A3, 2021.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 268-6

High-Order Mesh Generation and Mesh Adaptation for Complex Geometries with the Open-Source Code NekMesh

Kaloyan S. Kirilov[1,*], Joaquim Peiró[1], Jingtian Zhou[1], Mashy D. Green[2], David Moxey[2]

[1] Department of Aeronautics, Imperial College London, South Kensington Campus, London, SW7 2AZ, United Kingdom, www.imperial.ac.uk/aeronautics/
kaloyan.kirilov19@imperial.ac.uk, j.peiro@imperial.ac.uk, jingtian.zhou22@imperial.ac.uk,

[2] Department of Engineering, King's College London, Strand, London, WC2R 2LS, United Kingdom, www.kcl.ac.uk/engineering
mashy.green@kcl.ac.uk, david.moxey@kcl.ac.uk

Key Words: *High-Order, Curvilinear, Mesh Generation, Mesh Adaptation, hp-adaptation, Spectral/hp Element Methods*

High-fidelity spectral/hp methods possess superior diffusion and dispersion properties than conventional CFD methods and, potentially, better computational efficiency that makes them attractive for the modelling of transient flows about complex industrial geometries. However, these methods rely on high-quality curvilinear boundary-conforming meshes. However, the generation of such meshes for complex geometries is a significant barrier for the industrialisation of the methods, especially when highly stretched meshes are required to model high-Reynolds number viscous flow in the thin boundary layers adjacent to body surfaces.

To tackle this problem, we propose an alternative mesh generation workflow, where the complexity of generating the linear straight-sided mesh is offloaded to a third-party mesh generator and then all high-order activities are performed in NekMesh. This approach allows us to generate curved meshes with the same quality as that of the classical high-order bottom-up approaches [1,2,3], but takes advantage of the superior flexibility, robustness and user-friendly graphical user interface (GUI) of a state-of-the-art finite volume mesh generator.

We present first the details of a novel method for an efficient reconstruction of the B-Rep parameterization from the third-party straight-sided mesh that lacks the links to the CAD information needed for the generation of the high-order mesh. We then describe the state-of-the-art techniques employed to generate the high-order, namely mesh curving through projections, surface and volume optimizations with mesh sliding on the CAD B-Rep. We also discuss the use of high-order mesh modification techniques for improving the overall mesh quality in the context of hybrid unstructured meshes. Finally, we present some preliminary results on the application of *a posteriori* mesh adaptation procedure based on these mesh modification techniques.

Examples of application involving automotive geometries, such as the SSECOID project automotive test cases, an Inverted Multi-Element Wing and the full Imperial Front Wing [4], will be used to showcase the capabilities of the proposed high-order mesh generation and adaptation workflow and demonstrate the improved quality of the meshes it generates.

References

[1] M. Turner, J. Peiró, and D. Moxey. Curvilinear mesh generation using a variational framework. ComputerAided Design, 103:73–91, 2018.

[2] S. Chun, J. Marcon, J. Peiró, S. J. Sherwin, Reducing errors caused by geometrical inaccuracy to solve partial differential equations with moving frames on curvilinear domain, Computer Methods in Applied Mechanics and Engineering, Volume 398, 2022

[3] M. D. Green, K. S. Kirilov, M.Turner, J. Marcon, J. Eichstäedt, E. Laughton, C. D. Cantwell, S. J. Sherwin, J. Peiró, D. Moxey , NekMesh: An Open-Source High-Order Mesh Generation Framework. Available at SSRN: https://ssrn.com/abstract=4632754 or http://dx.doi.org/10.2139/ssrn.4632754

[4] F. F. Buscariolo, J. Hoessler, D. Moxey, A. Jassim, K. Gouder, J. Basley, Y. Murai, G. R.S. Assi, and S. J. Sherwin. Spectral/hp element simulation of flow past a formula one front wing: Validation against experiments. Journal of Wind Engineering and Industrial Aerodynamics, 221:104832, 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portuga

STS 268-7

Study on the Effect of the Upcoming Turbulent Boundary Layer on Automotive Test Case

Andrew O'Sullivan[1,*], Kunal Puri[1,2], Esteban Ferrer[3]

[1] Cadence Design Systems, Chaussée de la Hulpe 187, 1170 Brussels, Belgium
oandrew@cadence.com, kpuri@cadence.com

[3] ETSI School of Aeronautics, Universidad Politécnica de Madrid, Plaza Cardenal Cisneros 3, E-28040 Madrid, Spain, esteban.ferrer@upm.es

Key Words: *Turbulent boundary layer, automotive test cases, Large Eddy Simulation (LES)*

In the present study, we plan to compare different strategies to simulate the upcoming turbulent boundary layer for LES codes on automotive test cases and quantify the impact. The test case is the yawed Windsor Body which is one of the two automotive benchmarks used for AutoCFD 3 and AutoCFD4 workshop [1] and which has extensive experimental and previous CFD results to compare against. The upcoming turbulent boundary layer is of similar size as the gap between the bottom of the Windsor Body and the floor, making it thus a test case of interest to verify the sensitivity to the upcoming boundary layer. For the study, we will use a low Mach, Finite Volume Wall Modeled Large Eddy Simulation solver together with the turbulence generator strategies. We plan to run the test case not only at the operating condition of the AutoCFD workshop but verify the sensitivity to the upcoming boundary layer thickness and the inlet turbulence intensity. We will analyse the effect on : the mean forces (drag and side force), the variance of the forces, frequencies of the force oscillations, wake structures and Cp profiles.

The goal is to understand which metrics are more sensitive to the CFD inlet settings such that this can be applied to other automotive test cases where the operating conditions (upcoming boundary layer) are either unknown or incomplete; and thus determine whether the upcoming boundary layer can explain the differences with experimental results on other test cases.

Acknowledgement
This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grand agreement No 955923.

References

[1] Oxford University. *4THAUTOMOTIVECFDPREDICTIONWORKSHOP*. https://autocfd.eng.ox.ac.uk/.

[1] Early Stage Researcher in SSeCoID Project, R&D Brussels, oandrew@cadence.com (link for backup #)
[2] Software Engineering Director, R&D Brussels, AIAA Senior Member, kpuri@cadence.com,
[3] Professor in Applied Mathematics, School of Aeronautics, esteban.ferrer@upm.es

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 268-8

Transonic Buffet Prediction around Laminar Airfoils Utilizing Linear Global Stability Analysis

Matthias Plath[1,*], Florent Renac[1,2], Olivier Marquet[3] and Christian Tenaud[4]

[1] DAAA, ONERA, Université Paris Saclay, 92322 Châtillon, France,
[1]matthias.plath@onera.fr, [2] florent.renac@onera.fr

[3] DAAA, ONERA, Université Paris Saclay, 92190 Meudon, France, olivier.marquet@onera.fr

[4] Université Paris-Saclay, CNRS, CentraleSupélec, Laboratoire EM2C, 91190 Gif-sur-Yvette, France, christian.tenaud@centralesupelec.fr

Key Words: *Transonic buffet, transition model, global stability analysis*

Experimental [1] and numerical studies [3, 2] of transonic flows around airfoil have shown that the shock unsteadiness (also called buffet) is altered by the laminar or turbulent state of the incoming boundary layer. By performing LES simulations and SPOD of the transonic flow around an OALT25 airfoil, Zauner et al. [2] showed more specifically the co-existence of a low-frequency buffet mode and an intermediate frequency laminar separation bubble mode. The present work aims to investigate the onset of such unsteady phenomena by conducting a global stability analysis of the transonic steady flow within a RANS framework. Towards that objective, we consider the Spalart-Allmaras model (SAneg) as a turbulence model together with a γ transition model. The study reveals the presence of an unstable low frequency eigenmode, whose frequency and stable/unstable regions coincide very well with the low frequency buffet reported by Zauner et al. [2]. Further, performed unsteady simulations (URANS) resemble the complex dynamics of multiple shock waves and flow separation, in good agreement to the LES findings. The study will be further extended for higher Reynolds numbers to assess the accuracy of the proposed RANS global stability framework for laminar transonic buffet prediction.

References

[1] Brion, V. and Dandois, J. and Abart, J.-C. and Paillart, P., Experimental Analysis of the Shock Dynamics on a Transonic Laminar Airfoil. *Progress in Flight Physics –*, Vol **9**, pp. 365–386, 2017

[2] Zauner, M. and Moise, P. and Sandham, N. D., On the Co-existence of Transonic Buffet and Separation-Bubble Modes for the OALT25 Laminar-Flow Wing Section. *Flow Turbulence Combust.*, Vol **110**, pp. 1023–1057, 2023

[3] Zauner, M. and Sandham, N. D., Modal Analysis of a Laminar-Flow Airfoil under Buffet Conditions at Re = 500,000. *Flow Turbulence Combust.*, Vol **104**, pp. 509–532, 2020

STS 269 A & B

Engineering the Future: Advancements in Industrial Aerodynamic Simulations

Chair: Oriol Lehmkuhl

Barcelona Supercomputing Center, Plaça Eusebi Güell, 1-3, 08034 Barcelona, Spain
oriol.lehmkuhl@bsc.es, https://www.bsc.es

Session Abstract

Key words: *FVM/FE, SEM/FR/DG, LES, AMR, aeronautical applications, GPUs, extreme scaling*

The aviation sector presently accounts for over 2% of global greenhouse gas (GHG) emissions, emissions from international aviation could surge nearly fourfold by 2050 compared to 2010. It is a collective responsibility of the scientific community and the aeronautical industry to pioneer novel, more efficient designs. In the pursuit of enhanced efficiency and environmental responsibility, numerical simulation techniques, such as Computational Fluid Dynamics (CFD), are emerging as pivotal tools in aeronautical design. These tools will be the linchpin differentiating success from failure. Nevertheless, despite the current integration of CFD in the design process, there exists a pressing need to elevate the capabilities of existing numerical simulation tools tailored for aeronautical design. This entails a transformative shift toward re-engineering these tools to harness the power of extreme-scale parallel computing platforms. This strategic transition will empower the aerospace industry to leverage High-Performance Computing (HPC) within the design loop, a crucial step toward achieving the performance and environmental objectives outlined in the European Union's ambitious targets.

The present special technology session (STS) will be focused on the efforts to address this challenge. The topics of interest are related to improving the convergence of current numerical algorithms; adaptive mesh refinement algorithms; increase the maturity of HoM; overcome barriers to achieving extreme scale computing (load balancing, communication patterns, GPU integration, etc.); development of algorithms for data management, visualisation and modelling and benchmarking of large-scale aeronautical applications.

Contributing papers to the session parts A and B:
Part A: STS 269-1 to STS 269-6; **Part B:** STS 269-7 to STS 269-12.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 269-1

Cost Effective Wall-Modeled Large Eddy Simulations for Aeronautical Flow Analysis

Aitor Amatriain[1,2]*, Corrado Gargiulo[2], and Gonzalo Rubio[2,3]

[1] CERN, Geneva 23, 1211, Switzerland.
[2] E.T.S. Ingenieria Aeronautica y del Espacio, Universidad Politécnica de Madrid, Plaza Cardenal Cisneros 3, 28040 Madrid, Spain.
[3] Center for Computational Simulation, Universidad Politécnica de Madrid, Campus de Montegancedo, Boadilla del Monte, 28660 Madrid, Spain, aitor.amatriain@cern.ch

Key Words: *Computational fluid dynamics, turbulence modeling, aeronautical flow*

In this work, numerical simulations are conducted using the recently developed generalized wall-modeled large eddy simulation (GWMLES) model. It consists of an extended formulation of the classical WMLES approach, which also enables the modeling of the entire log-layer by using a Reynolds-Averaged Navier Stokes (RANS) model [1]. It provides a level of accuracy similar to recent LES results, as well as computational cost savings that are proportional to the Reynolds number in wall bounded flows. Considering standard benchmark cases with features of industrial flows, GWMLES has been shown to reduce the drawbacks of current methodologies that are based on near-wall modeled turbulence [2, 3], representing a significant advancement in the simulation of wall-bounded flows at high Reynolds numbers. The current contribution reviews the main features and outcomes of the newly developed GWMLES approach, and showcases its power with a complex industrial case that is representative of an aeronautical flow. The comparison uses extensive LES and experimental data as references, with a particular focus on spectral analysis.

References

[1] A. Amatriain, C. Gargiulo, and G. Rubio. Generalized wall-modeled large eddy simulation model for industrial applications. Physics of Fluids 36(015141), 2024.

[2] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. K. Strelets, and A. Travin. A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities. Theoretical and Computational Fluid Dynamics 20:181–195, 2006.

[3] S. T. Bose and G. I. Park. Wall-Modeled Large-Eddy Simulation for Complex Turbulent Flows. Annual Review of Fluid Mechanics 50:535–561, 2018.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 269-2

Efficient and Robust Implicit Solvers for Unsteady Flow Problems Using Harmonic Balance

Christian Frey[1]*, Graham Ashcroft[2] and Jan Backhaus[3]

Institute of Propulsion Technology, DLR - German Aerospace Center,
Linder Höhe, 51147 Cologne, Germany, www.dlr.de/en/at/
[1] christian.frey@dlr.de, [2] graham.ashcroft@dlr.de, [3] jan.backhaus@dlr.de

Key Words: *Harmonic balance, turbomachinery, CFD, implicit solver, pseudo time marching*

The simulation of time-periodic unsteady flows is a central problem in aeronautical applications, especially in turbomachinery. The so-called *harmonic balance* method which uses a spectral discretisation of the time-derivative has been shown to be a highly efficient approach for applications in unsteady aerodynamics and nonlinear aeroelasticity [2,1]. In contrast to linearised frequency-domain methods, harmonic balance takes the nonlinear interaction between harmonics into account.

In this paper we discuss various solution methods for the harmonic balance equation systems. Our main focus is on implicit pseudo time marching methods and iterative linear solvers. It turns out that the choice of an appropriate method depends on the configuration and the degree of nonlinearity involved. We show that one can, in many situations, simplify the linear implicit systems, thereby reducing the computational costs considerably. The results, however, also indicate that such simplification come at the expense of robustness in off-design conditions (e.g. separated flow) in combination with high amplitude disturbances. This implies that an efficient overall simulation strategy amounts to applying simplifying assumptions where applicable, while the robust and computationally more costly variant should only be used for highly non-linear phenomena.

References

[1] C. Berthold, J. Gross, C. Frey, and M. Krack. “Development of a fully-coupled harmonic balance method and a refined energy method for the computation of flutter induced Limit Cycle Oscillations of bladed disks with nonlinear friction contacts”. In: *Journal of Fluids and Structures* 102 (2021). doi: 10.1016/j.jfluidstructs.2021.103233.

[2] K. C. Hall, J. P. Thomas, and W. S. Clark. “Computation of Unsteady Nonlinear Flows in Cascades Using a Harmonic Balance Technique”. In: *AIAA J.* 40.5 (2002), pp. 879–886. doi: 10.2514/2.1754.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 269-3

Low Mach Preconditioning for Harmonic Balance Solvers

Pierre Sivel[1]* and Christian Frey[2]

Institute of Propulsion Technology, DLR - German Aerospace Center,
Linder Höhe, 51147 Cologne, Germany, www.dlr.de/en/at/
[1] pierre.sivel@dlr.de, [2] christian.frey@dlr.de

Key Words: *Computational Fluid Dynamics, Low Mach Preconditioning, Harmonic Balance*

Density-based solvers, originally designed for high Mach number flows, suffer from a decrease in solution quality and convergence rate as the Mach number decreases. Low Mach preconditioning has been shown by previous investigations [1] to improve the efficiency of density-based solvers at low Mach numbers in steady-state computations. Applying a similar procedure, this study presents the implementation of low Mach preconditioning in harmonic balance solvers to enhance the solution quality and optimize the convergence rate of unsteady frequency-domain computations in the low Mach regime.

The convergence rate is improved by multiplying the time derivative of the governing equations with a preconditioning matrix, to equalizes the acoustic and convective propagation velocities. However, for unsteady equations, the preconditioner introduces a frequency dependence in the acoustic propagation velocity. Thus, to optimize the convergence rate in the Harmonic Balance solver, each harmonic requires its own frequency dependent preconditioner. Inaccuracies at low Mach numbers occur due to the excessive numerical dissipation of the roe solver. In this study we utilize a preconditioned Roe solver for unsteady computations [2], which ensures the correct Mach number scaling in the artificial diffusion, significantly improving the solution quality.

Using the low Mach preconditioned Harmonic Balance solver, we computed a laminar cylinder and found that an inappropriate preconditioner can reduce the convergence rate. Optimal convergence is attained by employing the steady-state preconditioner for the zeroth harmonic and applying the frequency-dependent preconditioner for higher harmonics. In addition, the study highlights the shortcomings of the standard Roe solver in predicting the solution's unsteadiness for decreasing Mach numbers. In contrast, the preconditioned Roe solver produces Mach number independent solutions, establishing its superior predictive capability.

References

[1] Sivel, P. and Frey, C. and Kügeler, E. and Keil, M. Low Mach preconditioning for turbomachinery flow simulations with cavities and variable gas compositions. ASME Turbo expo 2022.

[2] Potsdam, M. and Sankaran, V. and Pandya, S. Unsteady Low Mach Preconditioning with Application to Rotorcraft Flows. 18th AIAA Computational Fluid Dynamics Conference. 2007

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 269-4

Parallel Performance of SOD2D: A High-Order Spectral Element Code

Jordi Muela[1]*, Lucas Gasparino[2] and Oriol Lehmkuhl[3]

Barcelona Supercomputing Center (BSC), Placa Eusebi Guell 1-3, 08034 Barcelona, Spain,
[1]jordi.muela@bsc.es, [2] lucas.gasparino@bsc.es, [3] oriol.lehmkuhl@bsc.es, www.bsc.es

Key Words: *Spectral Methods, Computational Fluid Dynamics, Continuous Galerkin method, Parallel CFD applications, Exascale computing, GPU parallelization*

This work presents the parallel performance of the SOD2D code (**S**pectral high-**O**rder co**D**e **2** solve partial **D**ifferential equations) [1], a new Continuous Galerkin High-Order Spectral Element Method (CG-SEM) code designed to solve simulations of both turbulent compressible and incompressible flows.

The current trend in new High-Performance Computing (HPC) platforms is to achieve higher computing performances by installing increasingly more modules based on GeneralPurpose Graphical Processing Units (GPGPUs) [2]. Hence, the new scientific computing codes must be programmed in such a way that they can benefit from the fine-grained parallelism and large data throughput of these devices. SOD2D has been coded following this new paradigm, and the present work aims to assess and analyse the parallel performance and scalability of SOD2D code, focusing on when running on GPUs.

Preliminary results show an almost ideal parallel efficiency in the weak speedup for all the computed GPU load ratios, in both compressible and incompressible solvers. Regarding the strong speedup analysis, the results are also very good, although these are strongly affected by the GPU load. The strong speedup is highly degraded in cases where the load per GPU is small and the graphical unit is not performing at its optimal point. Furthermore, the performance of SOD2D when using MPI versus NCCL (NVIDIA Collective Communications Library) implementations is analysed and compared, showing the potential and limitations of each one.

References

[1] L. Gasparino, F. Spiga, and O. Lehmkuhl, “Sod2d: A GPU-enabled spectral finite elements method for compressible scale-resolving simulations,” *Computer Physics Communications*, vol. 297, p. 109067, 2024.

[2] A. Tekin, A. Tuncer Durak, C. Piechurski, D. Kaliszan, F. Aylin Sungur, F. Roberts´en, and P. Gschwandtner, “State-of-the-art and trends for computing and interconnect network solutions for HPC and AI,” *Partnership for Advanced Computing in Europe, Available online at www.prace-ri.eu*, 2021.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 269-05

On the Extension of the Entropy Viscosity Method for Scale Resolving Incompressible Flows

Oriol Lehmkuhl[1]*, Lucas Gasparino[2] and Jordi Muela[3]

Barcelona Supercomputing Center (BSC), Calle de Jordi Girona 31, 08034 Barcelona, Spain,
[1] oriol.lehmkuhl@bsc.es, [2] lucas.gasparino@bsc.es, [3] jordi.muela@bsc.es, www.bsc.es

Key Words: *High-order finite element method, scale resolving methods, low-dissipation schemes, fluid mechanics.*

This work will present a GPU-capable low-dissipation semi-implicit SEM algorithm suitable for performing scale-resolving simulations of wall-bounded turbulent incompressible flows at high Reynolds. Such a model is motivated by issues encountered when performing LES and DNSs of typical industrial flows, where small and skew element, especially for the high-order types of elements used in SEM, can introduce several stability problems even when using the latest in supercomputing hardware.

To achieve this objective, the open-source code e SOD2D code (Spectral high-Order coDe 2 solve partial Differential equations [1], a new Continuous Galerkin High-Order Spectral Element Method (CG-SEM), is extended to incompressible flows, using a high-order/splitting velocity correction algorithm [2] together with a novel extension of the entropy viscosity method [3]. The resulting low dissipation scheme works will with high-order SEM, physical based SGS and WMLES models, and is able to be stable and accurate in complex geometries. The final work will detail the new stabilisation scheme and will discuss out several verification exercises at relevant Reynolds conditions.

References

[1] L. Gasparino, F. Spiga, and O. Lehmkuhl, "Sod2d: A GPU-enabled spectral finite elements method for compressible scale-resolving simulations", Computer Physics Communications, vol. 297, p. 109067, 2024

[2] G.E. Karniadakis, M. Israeli, S.A. Orszag, "High-Order Splitting Methods for the Incompressible Navier-Stokes Equations". Journal of Compuational Physics, 414-443. 1991.

[3] Jean-Luc Guermond, Richard Pasquetti, Bojan Popov, "Entropy viscosity method for nonlinear conservation laws", Journal of Computational Physics, Volume 230, Issue 11, 2011, Pages 4248-4267.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 269-6

On a Semi-Implicit Entropy Viscosity SEM Algorithm for Wall-Bounded Compressible LES and DNS

Lucas Gasparino[1]* and Oriol Lehmkuhl[2]

Barcelona Supercomputing Center (BSC), Calle de Jordi Girona 31, 08034 Barcelona, Spain,
[1]lucas.gasparino@bsc.es, [2] oriol.lehmkuhl@bsc.es, www.bsc.es

Key words: *SEM, LES, DNS, IMEX, compressible flow, GPU computing*

This work will present a GPU-capable low-dissipation semi-implicit SEM algorithm suitable for performing scale-resolving simulations of wall-bounded, turbulent compressible flows, including at supersonic regimes. Such a model is motivated by issues encountered when performing LES and DNSs of typical aeronautical flows, where small element sizes, especially for the high-order types of elements used in SEM, imply that the maximum time-step size possible with an explicit method quickly becomes unfeasible even when using the latest in supercomputing hardware.

Geometry restrictions in aeronautical bodies, such as at wing tips or trailing edges, impose hard limits on how large a cell or element in a discretized mesh can be. High-order elements have their minimal edge size determined by their smallest two-point segment, which can be significantly smaller than the edge size for very high polynomial orders, a situation that is even worse for SEM, where nodes closer to the element corners are more tightly packed. Flow conditions in these regions can dictate minuscule time-step sizes determined by the diffusive Courant condition, undermining the use of current supercomputing systems.

Semi-implicit schemes propose an interesting solution to this issue: for the compressible flow equations, the stiff terms that need to be treated implicitly are either linear or quasi-linear, leading to a much simpler implicit model that does not depend on a linearization procedure like Newton's method, while complex nonlinear terms are dealt with by the explicit part, which presents no computational complexity when compared to an implicit counterpart. Stability limits for these schemes lead to a time-step size that depends on the convective Courant number, which is far less restrictive for small element sizes.

We focus on the IMEX family of schemes, an explicit/implicit form of Runge-Kutta time-marching algorithm capable of using separate Butcher tables. Based on the work presented in [1], we select a suitable IMEX scheme to couple with the entropy viscosity spatial discretization described in [2], proceeding to showcase its stability and performance on accelerated hardware based on our own implementation using OpenACC.

References

[1] D. Cavaglieri, T. Bewley, Low-storage implicit/explicit Runge–Kutta schemes for the simulation of stiff high-dimensional ODE systems, J. Comput. Phys. 286 (2015), 172-193

[2] L. Gasparino, F. Spiga, O. Lehmkuhl, SOD2D: A GPU-enabled Spectral Finite Elements Method for compressible scale-resolving simulations, Comput. Phys. Commun. 297 (2024)

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 269-7

HAWT Efficiency Increase via Active Flow Control Implementation

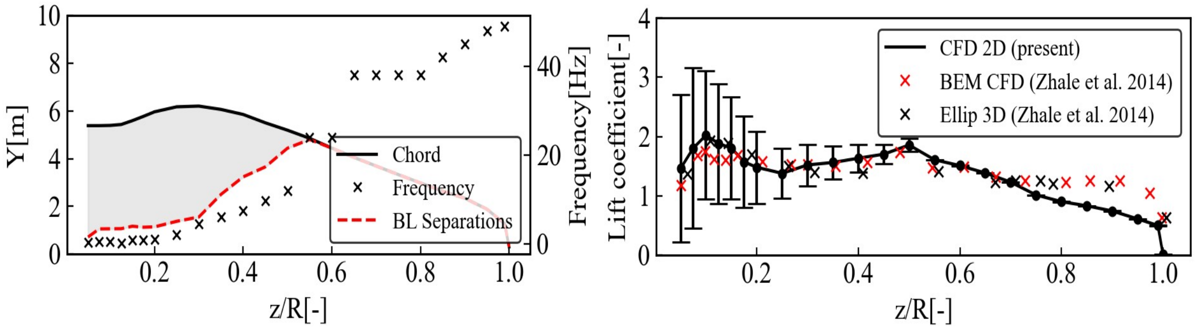
Ahmad Nabhani[1]*, Mohammad Saemian[1] and Josep M Bergada[1,2]

[1] Universitat Politécnica de Catalunya – BarcelonaTech (UPC), Barcelona, Spain,
[2]bergada@mf.upc.edu, www.upc.edu/en/

Key Words: *CFD, Finite Volume Method, Aerodynamic, Horizontal Axis Wind Turbine, Boundary Layer, Active Flow Control, Synthetic Jet*

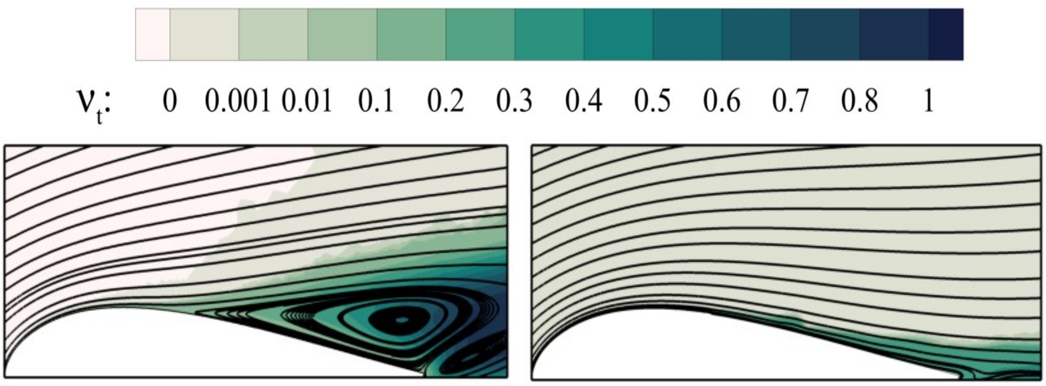
In this research, we conducted a comprehensive Computational Fluid Dynamics (CFD) investigation on the DTU-10MW reference blade wind turbine [1], employing the Unsteady Reynolds-Averaged Navier-Stokes (URANS) $K - \omega$ SST turbulence model. In general, there is a lack of experimental data for large horizontal axis wind turbines (HAWT), and most of the existing studies have predominantly relied on potential-flow methods, Blade Element Momentum (BEM) methods, and vortex methods [2]. These approaches fail to provide a good understanding of the boundary layer (BL) dynamics around the turbine blades, particularly in pinpointing separation points.

Our study addresses this gap by performing a series of 2D simulations on 24 sections cut along the turbine blade. This allows us to identify for each section, the BL thickness and its time-averaged separation point, the dynamic lift and drag coefficients, and its associated oscillation amplitude. The vortex shedding frequency, pressure, and skin friction coefficients were as well obtained for each section. Some of the results for a wind speed of 10*m/s* are presented in Figure 1, where the BL time-averaged separation point, its associated frequency, the lift coefficient, and its peak-to-peak amplitude are introduced as a function of the blade non-dimensional radius. A good agreement is observed with the previous research undertaken by [3]. This initial information is needed to implement the Active Flow Control (AFC) technology in the sections where BL separation exists. After optimizing the AFC parameters and when employing Synthetic Jets (SJ) on the blade section $z/R = 0.35$, the resulting CFD simulation shows that the boundary layer was almost fully reattached, this can be seen in Figure 2 where the streamlines on the upper airfoil surface for the baseline case and the AFC one are presented. In order to evaluate the WT efficiency increase, the AFC implementation in the section studied was energy assessed, it was obtained that the SJAFC system consumed only 6.4% of the increase of energy generated by the wind turbine, therefore highlighting the potential of AFC techniques to enhance HAWT's efficiency. At the moment we are performing the optimization of the five associated AFC parameters for all sections where the BL is separated, once the CFD simulations with the corresponding optimized parameters will be done, the maximum HAWT overall power gain will be obtained. The methodology here introduced can be applied to any HAWT and under any operating condition, in order to evaluate the maximum energy increase a given WT can generate.



(a) **(b)**

Figure 1: *(**a**) B.L. separation points along the chord versus the blade. (**b**) Comparison of Lift coefficient distribution along the blade (Error bars showing the amplitude of vortex shading) with previous works [3].*



(a) **(b)**

Figure 2: *Streamlines of the temporal average velocity field and contour of turbulent kinematic viscosity. (**a**) baseline case at z/R=0.35, (**b**) AFC case.*

References

[1] C. Bak, F. Zahle, R. Bitsche, T. Kim, A. Yde, L. C. Henriksen, A. Natarajan, and M. Hansen, "Description of the dtu 10 mw reference wind turbine–dtu wind energy report-i-0092," *Technical University of Denmark, Fredericia (Denmark)*, 2013.

[2] S. G. Horcas, F. Debrabandere, B. Tartinville, C. Hirsch, and G. Coussement, "Rotortower interactions of dtu 10mw reference wind turbine with a non-linear harmonic method," *Wind Energy*, vol. 20, no. 4, pp. 619–636, 2017.

[3] F. Zahle, C. Bak, N. N. Sørensen, S. Guntur, and N. Troldborg, "Comprehensive aerodynamic analysis of a 10 mw wind turbine rotor using 3d cfd," in *32nd ASME Wind Energy Symposium*, p. 0359, 2014.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 269-8

Deep Learning Surrogate Models for the Automotive Industry

Benet Eiximeno[1,2]*, Arnau Miró[1,2], Ivette Rodríguez[2] and Oriol Lehmkuhl[1]

[1] Barcelona Supercomputing Center, 08034 Barcelona, Spain, www.bsc.es
benet.eiximeno@bsc.es, arnau.mirojane@bsc.es, oriol.lehmkuhl@bsc.es,

[2] Turbulence and Aerodynamics Research Group. Universitat Politècnica de Catalunya (UPC), 08221 Terrassa, Spain, Ivette.rodriguez@upc.edu

Key Words: *Reduced order models (ROM), deep learning, aerodynamics, variational autoencoders*

When a car is exposed to lateral wind gusts, the yaw angle of the incident velocity increases, leading to an intensification of the suction at the vehicle's rear and elevated drag forces. This effect is particularly relevant in square-back vehicles, as their drag force is completely driven by the pressure on their back face. This work focuses on building a surrogate model for the changes in the aerodynamic performance with the yaw angle of a simplified square-back vehicle, the Windsor body.

The training (δ = [2,5°, 5° and 10°]) and validation datasets (δ = 7.5°) have been generated with wall-modeled large eddy simulations at a Reynolds number of $Re = 2.9 \times 10^6$. The simulations have been performed using SOD2D [1] and the obtained results are in agreement with the experiments from Varney et al. [2].

A variational autoencoder [3] with two latent vectors has been trained with the data of the back pressure showing that the values in the latent space for the mean back pressure depend linearly on the yaw angle. Furthermore, when a set of time-correlated snapshots are passed through the encoder, the latent vector retains the dominant frequencies of the most relevant DMD [4] modes of the system. With these properties, it has been possible to compute the mean back pressure at δ = 7.5° with a maximum deviation of 3.1%.

In the final version of this work, the extrapolation of the model outside the training range and its performance on the wake velocity will also be discussed.

References

[1] L. Gasparino, F. Spiga, and O. Lehmkuhl, "Sod2d: A GPU-enabled spectral finite elements method for compressible scale-resolving simulations," *Computer Physics Communications*, vol. 297, p. 109067, 2024.

[2] M. Varney, G. Pavia, M. Passmore, and C. Crickmore, "Windsor model experimental aerodynamic dataset," 2020.

[3] D. P. Kingma and M. Welling, "Auto-encoding variational bayes", *arXiv preprint arXiv:1312.6114*, 2013.

[4] P. J. Schmid, "Dynamic mode decomposition of numerical and experimental data," *Journal of fluid mechanics*, vol. 656, pp. 5–28, 2010.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 269-9

Adaptive Mesh Refinement for Spectral-Element Simulations of Compressible Turbulent Flows

Samuel Gómez[1]*, Jordi Muela[2], Abel Gargallo[3] and Oriol Lehmkuhl[4]

Barcelona Supercomputing Center (BSC), Placa Eusebi Guell 1-3, 08034 Barcelona, Spain,
[1]samuel.gomez@bsc.es, [2] jordi.muela@bsc.es, [3] abel.gargallo@bsc.es, [4] oriol.lehmkuhl@bsc.es,
www.bsc.es

Key Words: *Spectral element method (SEM), Adaptive mesh refinement (AMR), computational fluid mechanics (CFD), large eddy simulations (LES)*

The modern landscape is characterized by a growing concern for emissions and energy efficiency, with transportation being a focal point. In this context, the aerodynamic force holds particular significance in influencing vehicle dynamics. Effectively addressing and mitigating this force requires the utilization of reliable tools for design enhancement, and Computational Fluid Dynamics (CFD) plays a crucial role, especially with the incorporation of accelerated processing through Graphics Processing Units (GPU).

In order to solve this kind of flows, we use a combination of Spectral Finite Elements, operator splittings on the convective term, and the Entropy Viscosity stabilization model on the GPU accelerated code SOD2D [1].

Given the inherently turbulent nature of these flows, characterized by temporal and spatial multi-scale attributes, a proper discretization strategy is essential. For time integration, we use a fourth-order Runge-Kutta. In space, we start with a coarse mesh and that is later adapted based on the problem solution [2]. As SEM performance benefits from full-hexadral meshes the adaptation process utilizes an octree-based structure, resulting in non-conformity across element faces. Consequently, enforcing continuity across these interfaces becomes crucial. We choose to impose the solution from the fine interface to the coarse one and the residual in the opposite case in order to address this issue. Initial implementations of this approach demonstrate minimal loss of mass and energy for the Taylor-Green vortex case reduced as the polynomial order is increased. More complex cases as the High Lift Prediction and AutoCFD geometries will be explored.

References

[1] Gasparino, L., Spiga, F., and Lehmkuhl, O. (2023). SOD2D: A GPU-enabled Spectral Finite Elements Method for compressible scale-resolving simulations. Computer Physics Communications, 109067.

[2] Fischer, P. F., Kruse, G. W., and Loth, F. (2002). Spectral element methods for transitional flows in complex geometries. Journal of Scientific Computing, 17, 81-98.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS269-10

Towards Active Flow Control Strategies through Deep Reinforcement Learning

Ricard Montala[1]*, Bernat Font[2], Oriol Lehmkuhl[3], Ricardo Vinuesa[4] and Ivette Rodriguez[1]

[1] Universitat Politècnica de Catalunya (UPC), 08222 Terrassa, Spain,
ricard.montala@upc.edu, ivette.rodriguez@upc.edu

[2] Delft University of Technology (TU Delft), 2629 HS Delft, Netherlands, B.Font@tudelft.nl

[3] Barcelona Supercomputing Center (BSC), 08034 Barcelona, Spain, oriol.lehmkuhl@bsc.es

[4] KTH Royal Institute of Technology, 100 44 Stockholm, Sweden, rvinuesa@mech.kth.se

Key Words: *Computational fluid dynamics (CFD), Aerodynamics, Turbulence, Active flow control, Deep reinforcement learning*

The present work proposes to explore deep reinforcement learning (DRL) methods for designing optimal active flow control (AFC) strategies for drag reduction using synthetic jets in wings. In this manner, an agent built through an artificial neuronal network (ANN) finds the most optimal actuation based on the current state of the flow and previous experiences learned during the training process.

The first study that successfully applied DRL to AFC problems was Rabault et al. [1], which considered a 2D cylinder at a Reynolds number of Re = 100. Subsequent investigations implemented the methodology to 3D cylinders [2], maintaining the Reynolds number low. Other works also applied DRL and AFC to airfoils or wall-bounded flows, all considering weakly turbulent conditions. Therefore, the interest is placed now on integrating DLR and AFC to address more complex flow problems with turbulent conditions.

In the present study, DRL is applied to the SD7003 airfoil at Re = 60000, as previously investigated in Rodriguez et al. [3], who tested different AFC actuations with synthetic jets. To assess the capabilities of the methodology in reducing the drag coefficient in wings, the angles of attack studied are $\alpha = 4°$ and $14°$, obtaining promising results in both wall-bounded and massive separated conditions, respectively.

References

[1] J. Rabault, M. Kuchta, A. Jensen, U. Réglade and N. Cerardi. Artificial neuronal networks trained through deep reinforcement learning discover control strategies for active flow control. Journal of Fluid Mechanics, 856:281–302, 2019.

[2] P. Suárez, F. Alcántara-Avila, A. Miró, J. Rabault, B. Font, O. Lehmkuhl and R. Vinuesa. Active flow control for three-dimensional cylinders through deep reinforcement learning. ETMM14 proceedings, Barcelona, 2023.

[3] I. Rodriguez, O. Lehmkuhl and R. Borrell. Effects of the actuation on the boundary layer of an airfoil at Reynolds number Re=60000. Flow, Turbulence and Combustion, 105:607-626, 2020.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 269-11

Development and Exploitation of Scale-Resolving Simulation Tools for Turbomachine Flows

Michel Rasquin[1], Margaux Boxho[1], Olivier Coulaud[1], Manuel Diaz[1], Andrea Rocca[1], Koen Hillewaert[1,2] and Thomas Toulorge[1]*

[1] Cenaero, Rue des Frères Wright, 29, 6041 Gosselies, Belgium, www.cenaero.be
michel.rasquin@cenaero.be, thomas.toulorge@cenaero.be

[2] Aerospace and Mechanics Dept., Université de Liège, 4000 Liège, Belgium
Koen.Hillewaert@uliege.be

Key Words: *Scale-Resolving Simulation, Discontinuous Galerkin Method, High-Performance Computing, Mesh Adaptation, Statistical Convergence Estimator, Turbulence Modeling*

In response to the threat of climate change, ambitious targets have been defined at European level for the environmental performance of the aviation industry, namely dramatic reductions in greenhouse gas emissions by 2035 and climate neutrality by 2050. These challenging objectives can only be achieved through radical innovation in aircraft propulsion, including revolutionary engine architectures and propulsion systems based on alternative fuels. However, such changes in engine design imply the ability for aerodynamic simulation tools to explore a larger design space with a higher level of representativeness than current RANS-based CFD solvers are capable of. Scale-Resolving Simulation (SRS) methods, namely Large-Eddy Simulation (LES) and Direct Numerical Simulation (DNS), can significantly contribute to overcome the current barriers.

SRS is nowadays becoming applicable to turbomachine flows of industrial interest by means of mature high-order numerical methods in combination with new-generation supercomputers, that offer unprecedented computational power. In this talk, we will present on-going development efforts at Cenaero to build an SRS capability based on this principle, with emphasis on two aspects. The first one is the pursuit of high fidelity, both numerical (with high-order Discontinuous Galerkin method and accurate shock-capturing techniques) and physical (with representative inlet turbulence injection techniques). The second one is the efficient exploitation of modern High-Performance Computing (HPC) resources, with parallel mesh adaptation techniques, statistical convergence estimators and implementation on modern supercomputing architectures. Examples of applications will be given.

Despite the possibilities offered by the latest HPC systems, SRS methods remain too costly and specialized to be directly used for design optimization in industrial contexts, which limits their impact on turbomachine technologies. Another path to exploit the benefits of SRS is the use of LES and DNS results to benchmark, calibrate and improve turbulence models, particularly with the help of Machine Learning (ML) techniques. We will show how we employ SRS methods to generate high-fidelity turbulent flow databases and create new ML-based wall models.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 269-12

Efficient Algorithms for HPC in the New Generation Solver CFD ONERA-DLR-Airbus (CODA)

Vincent Couaillier[1,*], Riccardo Milani[2], Emeric Martin[3], Yongseok Jang[4], Jorge Nunez[5], Jean-Baptiste Chapelier[6] and Florent Renac[7]

ONERA, Univ. Paris-Saclay, *92322* Châtillon, France, www.onera.fr,
[1] vincent.couaillier@onera.fr, [2] riccardo.milani@onera.fr, [3] emeric.martin@onera.fr,
[4] yongseok.jang@onera.fr, [5] jorge.nunez@onera.fr, [6] jean-baptiste.chapelier@onera.fr,
[7] florent.renac@onera.fr

Key Words: *Computational Fluid Dynamics, Discontinuous Galerkin methods, Finite Volume methods, Numerical linear algebra, multigrid methods*

A major focus of the new aeronautical programs is the widespread use of numerical simulation in the design process. Industrial partners anticipate that numerical simulation requirements will strongly increase, possibly by an order of magnitude. Therefore, it is vital to reduce the computation times of simulation tools, particularly for aerodynamics, by relying on software architectures that are efficient in heterogeneous and flexible HPC environments in order to integrate new numerical models (accurate and efficient algorithms adapted to improved physical models).

In this context, Airbus, DLR and ONERA launched at the beginning of 2018 the development of the new generation CFD code CODA [1] to overcome the limits of current CFD tools. CODA is faster, more automatic and able of simulating complex aerodynamic [2] or Multiphysics flows. It integrates classical Finite Volume (FV) methods as well as new Discontinuous Galerkin (DG) methods that have been investigated for non-linear CFD [3] for nearly two decades in in-house codes and have been subjects of intensive evaluation in international workshops.

We will present the work carried out recently by ONERA during two collaborative projects: LAMA in a national framework, and NextSIM in a European framework (leaded by BSC). More specifically, we will present new developments on high-precision adaptive DG methods for improving the quality of physical results. This work is partly based on research carried out upstream in software prototypes and now benefits from the advanced HPC programming language of the CODA software. We will also show Reynolds-Averaged Navier-Stokes simulations leveraging efficient implicit resolution methods thanks to new developments on linear algebra preconditioning [4], as well as on multigrid methods adapted to unstructured multi-element grids based on optimized coarse grids generation.

Acknowledgments

The work presented here have received funding from the European Union's Horizon 2020 research and innovation program under grant agreement No. 956104 ("NextSim") or the French Directorate General for Civil Aviation (DGAC) project "LAMA".

References

[1] Görtz Stefan, Leicht Tobias, Couaillier Vincent, Méheut Michael, Larrieu Pascal, Champagneux Steeve, CODA: A European Perspective for a Next-Generation CFD, Analysis and Design Platform. NATO AVT-366 Workshop on Use of Computational Fluid Dynamics for Design and Analysis: Bridging the gap between Industry and Developers, 2022.

[2] Volpiani P. S., Chapelier, J. B., Schwöppe, A., Jägersküpper, J., & Champagneux, S. (2023). Simulating the Common Research Model using the new CFD software from ONERA, DLR and Airbus. In AIAA AVIATION 2023 Forum (p. 3275).

[3] Renac Florent, Entropy stable, robust and high-order DGSEM for the compressible multicomponent Euler equations. Journal of Computational Physics, 445 (2021), 110584.

[4] Jadoui Mehdi, Blondeau Christophe, Martin Emeric, Renac Florent, Roux François-Xavier, Comparative study of inner–outer Krylov solvers for linear systems in structured and high-order unstructured CFD problems. Computers & Fluids, Volume 244, 15 August 2022, 105575.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 271 A & B

High Aspect Ratio Wing Design and Development for Short- and Medium Range Aircraft

Chairs: Jos Vankan[1] and Bruno Stefes[2]

[1] Royal Netherlands Aerospace Centre NLR, Aerospace Vehicles Division,
Anthony Fokkerweg 2, 1059 CM Amsterdam, The Netherlands,
jos.vankan@nlr.nl, www.nlr.nl

[2] Airbus Operations GmbH, Flight Physics, Airbus Allee 1, 28199 Bremen, Germany,
bruno.stefes@airbus.com

Session Abstract

Key Words: *High aspect ratio wing, short- and medium range aircraft, drag reduction, optimized structures, numerical-experimental investigations, loads control.*

Sustainable aviation is a major challenge that requires technology developments in many different areas. One important area is the reduction of green-house gas (GHG) emissions, which is directly related to aircraft operations and energy efficiency. One of the key components for improving aircraft efficiency is drag reduction, and increased wing aspect ratio is a key enabler for that.

Therefore, this Special Technology Session (STS) focus on technologies for design and development of high aspect ratio (HAR) wings for short- and medium range (SMR) aircraft. This category of aircraft is responsible for a major contribution in aviation GHG emissions and is therefore important to address. At the same time, these aircraft have high-tech wings with advanced aerodynamics, optimized structures and complex integration of primary and secondary flight controls. The further improvement of these high-tech wings requires advanced modelling, innovative computational methods and design tools for all required technology areas. In particular, increasing the wing aspect ratio will require special attention for load control, for which combined numerical-experimental investigations are being pursued for example in the Clean Aviation UP Wing project.

The STS invited papers on the design, modelling, analysis, testing, validation, manufacturing and assembly of all the relevant technologies that are involved in the development of these advanced high-aspect ratio wings.

Acknowledgement

The project Ultra Performance Wing (UP Wing, project number: 101101974) is supported by the Clean Aviation Joint Undertaking and its members.

Contributing papers to the session parts A and B:

Part A: STS 271-1 to STS 271-6; **Part B:** STS 271-7 to STS 271-11;

Chairs' detailed affiliations:

Dr.ir. Jos Vankan
Principal Scientist
Royal Netherlands Aerospace Centre NLR
Aerospace Vehicles Division,
Collaborative Engineering Systems Departm.
Anthony Fokkerweg 2,
1059 CM Amsterdam, The Netherlands
T +31-88-5113059 | E jos.vankan@nlr.nl

Dr.-Ing. Bruno Stefes
Flight Physics
Airbus Operations GmbH,
Airbus Allee 1
28199 Bremen, Germany
Mobil: +49 172 4032258 bruno.stefes@airbus.com

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 271-1

Introduction to High Aspect Ratio Wing Design and Development for Short- and Medium Range Aircraft

Jos Vankan[1,*] and Bruno Stefes[2]

[1] Royal Netherlands Aerospace Centre NLR, Aerospace Vehicles Division, Anthony Fokkerweg 2, 1059 CM Amsterdam, The Netherlands, jos.vankan@nlr.nl, www.nlr.nl

[2] Airbus Operations GmbH, Flight Physics, Airbus Allee 1, 28199 Bremen, Germany, bruno.stefes@airbus.com

Key Words: *High aspect ratio wing, short- and medium range aircraft, drag reduction, optimized structures, numerical-experimental investigations, loads control*

Abstract:

Sustainable aviation is a major challenge that requires technology developments in many different areas. One important area is the reduction of green-house gas (GHG) emissions, which is directly related to aircraft operations and energy efficiency. One of the key components for improving aircraft efficiency is drag reduction, and increased wing aspect ratio is a key enabler for that.

Therefore, the Special Technology Session (STS) 271 will focus on technologies for design and development of high aspect ratio (HAR) wings for short- and medium range (SMR) aircraft. This category of aircraft is responsible for a major contribution in aviation GHG emissions and is therefore important to address. At the same time, these aircraft have high-tech wings with advanced aerodynamics, optimized structures and complex integration of primary and secondary flight controls. The further improvement of these high-tech wings requires advanced modelling, innovative computational methods and design tools for all required technology areas. In particular, increasing the wing aspect ratio will require special attention for load control, for which combined numerical-experimental investigations are being pursued for example in the Clean Aviation UP Wing project.

The STS271 will present papers on the design, modelling, analysis, testing, validation, manufacturing and assembly of all the relevant technologies that are involved in the development of these advanced high-aspect ratio wings.

This paper will present a general introduction into the subject of the STS271 and a global overview of technology developments in the relevant areas.

Acknowledgement:

The project Ultra Performance Wing (UP Wing, project number: 101101974) is supported by the Clean Aviation Joint Undertaking and its members.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-2

Aerodynamic Design of a High Aspect Ratio Strut-Braced Wing

Diego Losada Costoso[1]*, Eric Nguyen Van[2], Olivier Atinault[1,3] and Michael Méheut[1,4]
[1] ONERA, Université Paris-Saclay, 8 Rue des Vertugadins, 92190 Meudon, France,
[1]diego.losada_costoso@onera.fr, [3] olivier.atinault@onera.fr, [4] michael.meheut@onera.fr
[2] ONERA, Université de Toulouse, 2 Av. Edouard Belin, 31000 Toulouse, France,
eric.nguyen_van@onera.fr

Key Words: *strut-braced wing, aerodynamic design, airfoil, optimization, CFD*

As part of Clean Aviation's UP WING project, the present work aims at designing a hydrogen-powered transport aircraft for a Short-Medium Range mission (239 PAX, 2500 NM) while reducing its overall energy consumption compared to existing aircraft. From an aerodynamic point of view, one way to reduce drag is to use a higher aspect ratio wing [3]. The present work focuses on the transonic aerodynamic design of a strut-braced high-aspect-ratio wing. The study begins with the overall aircraft study, which provides a preliminary design with the general shape of the aircraft (positions and dimensions of the horizontal and vertical stabilizers, wing and fuselage), based on conceptual design assumptions. Then the design process uses the ESP/CAPS CAD modeler [1] and creates a geometry, which is used to assess the aerodynamic performance of the configuration (with automatic meshing, CFD RANS simulation and far-field drag analysis [4]). The aerodynamic wing design also includes an airfoil optimization loop. The preliminary design takes into account structural considerations coming from low fidelity models. In the final paper, the aerodynamic performance of the optimized shape will be analyzed in cruise and off-design conditions.

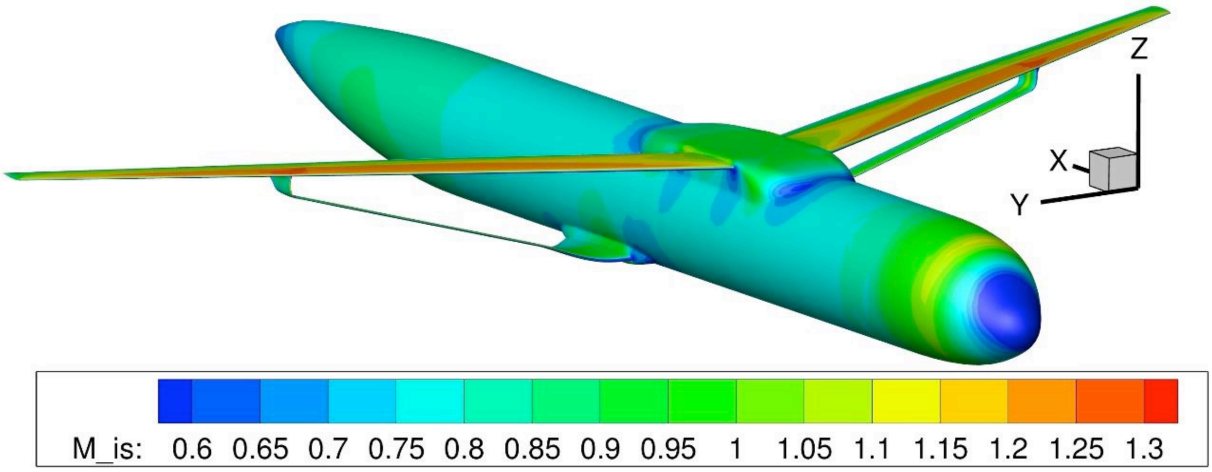


Fig. 1: *Isentropic Mach number on the strut-braced wing design in cruise conditions (M=0.78; Cl=0.65)*

References

[1] Haimes, R., Dannenhoffer, J., Bhagat, N. D., & Allison, "Multi-fidelity geometry-centric multi-disciplinary analysis for design", 2016 AIAA Modeling and Simulation Technologies

[2] G. Carrier, G. Arnoult, N. Fabbiane, J. Schotté, C. David, S. Defoort, M. Delavenne, E. Bénard "Multidisciplinary analysis and design of strut-braced wing concept for medium range aircraft", 2022 AIAA SciTech

[3] G. Carrier and al. "Investigation of A Strut-Braced Wing Configuration For Future Commercial Transport". ICAS 2012.

[4] Destarac, D. "Drag extraction from numerical solutions to the equations of fluid dynamics: the far-field philosophy." Maîtrise de la traînée et de l'impact sur l'environnement, 43ème Congrès d'Aérodynamique Appliquée de la 3AF (2008).

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 271-3

Structural Topology Optimization of a High-Aspect Ratio Dry-Wing for H2-Powered Aircraft

Cédric Julien[1,*], Lander Vertonghen[2] , Ludovic Coelho[3] and François-Xavier Irisarri[4]

[1] DMAS - Materials and Structures, ONERA, Univ. Paris-Saclay, 92322 Chatillon, France
[1,*] cedric.julien@onera.fr, [2] lander.vertonghen@onera.fr, [4]francois-xavier.irisarri@onera.fr

[4] DAAA - Aerodynamics Aeroelasticity Acoustics, ONERA, Univ. Paris-Saclay, 92322 Chatillon, France
[4] ludovic.coelho@onera.fr,

Key Words: *Structural optimization, topology optimization, composite materials, Hydrogen fuel*

Future environmental regulations put an increasingly stringent pressure on aircraft manufacturers to go even further in terms of fuel efficiency and reduction in carbon emissions. However, it is unlikely that the conventional "tube & wings" concept will manage to meet these requirements. As a result, innovative aircraft concepts have been increasingly studied, that rely on a combination of lightweight structures and novel energy sources, and feature the need to reevaluate the nature and positions of fuel storage, so that the wing itself no longer bears the function of carrying the fuel. The end result is that the design space of the wing itself expands substantially. New degrees of freedom emerge, from the internal structure of the wing, or the assembly to the fuselage, to the positioning and integration of movable parts and engines. In particular, when it comes to the internal structure of the wing, radically novel structural layouts become possible.

Precisely, the aim of this work is to exploit the potential of structural topology optimization to define new and interesting layouts for these future wing concepts. Indeed, a common practice for structural engineers in design offices is to try to find efficient shapes through the use of structural optimization methods. Among those, topology optimization allows to define a structure without prior knowledge or assumption on the shape or characteristics of the part [1]. Rather, the design variables are the very topology of the shape, usually characterized by density [2], as well as the mechanical properties of the material. In that regard, recent approaches have been able to extend this framework to anisotropic materials, as is typically the case for the CFRP that are used in the aeronautic industry, thus further increasing the performance of the optimized structures [3].

As such, the subject of this study is the high-aspect-ratio, strut-braced wing of a SMR commercial aircraft, powered with liquid H2. Starting from the sizing of this configuration using "state-of-the-art" methods for a conventional metallic wing-box structure for the wing, a first structural optimization is performed. The objective of this optimization is to achieve a best compromise between weight and a set of aeroelastic constraints, through the use of composite materials and the tailoring of their properties along the wing. The purpose of this first step is to identify a reference solution, which will serve as a basis for comparison with future designs.

Precisely, a second optimization step is then performed. In this step, a set of specific spars is selected from the conventional wing-box layout used in the first step. These spars may be carrying a higher level of loads, or they may be lying on critical locations in the wing, e.g. the connection with the engine mast or the strut. A topology optimization method dedicated to anisotropic materials is then applied to each of those domains, in order to find the most efficient layout possible when using composite materials. These optimized spars are then used in the calculation of the overall performance of the wing, and quantify the improvement with respect to the aforementioned reference solution.

To take this a step further and complete the process, future works will focus on extending this approach to the optimisation of the entire 3D volume of the wing. In particular, the potential of truss layout optimization will be investigated.

Acknowledgement
This work is part of the project Ultra Performance Wing (UP Wing, project number: 101101974), supported by the Clean Aviation Joint Undertaking and its members.

References
[1] Bendsøe M.P and Sigmund O, Topology Optimization, Springer Berlin Heidelberg, 2004.
[2] Sigmund O., A 99 line topology optimization code written in Matlab, Struct. Multidiscipl. Optim., 21 (2) (2001), pp. 120-127.
[3] Vertonghen L., Irisarri F.X., Bettebghor D. and Desmorat B., Gradient-based concurrent topology and anisotropy optimization for mechanical structures, Computer Methods in Applied Mechanics and Engineering, Volume 412, 2023.
[4] Opgenoord, M.M. and Willcox, K.E., Aeroelastic Tailoring using Additively Manufactured Lattice Structures, Multidisciplinary Analysis and Optimization Conference, Atlanta, 2018.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 271-4

High-Lift Challenges of High-Performance Wings

Guiseppe Mingione[1,*], Niko Bier[2], Luca Marino[3], Tobias Trautmann[4], Jurij Sodja[5], Michael Vellekoop[6], Pier Luigi Vitagliano[7], and Jochen Wild[8]

[1,7] CIRA, via Maiorise, 81043, Capua ITALY, [1] g.mingione@cira.it, [7] p.vitagliano@cira.it

[2,8] DLR, Institute of Aerodynamics and Flow Technology, Lilienthalplatz 7, 38108 Braunschweig, Germany, [2] Niko.Bier@dlr.de, [8] jochen.wild@dlr.de

[3,5] TU Delft, Kluyverweg 1, 2629 HS Delft, The Netherlands,
[3] L.Marino-1@tudelft.nl, [5] j.sodja@tudelft.nl

[4,6] IMSAS, University of Bremen, Otto-Hahn-Allee 1, 28359 Bremen, Germany,
[4] TTrautmann@imsas.uni-bremen.de, [6] MVellekoop@imsas.uni-bremen.de

Key Words: *Upwing, Clean-Aviation, High-Lift, Vortex Generators, Load alleviation*

Abstract

One of the key-elements for high performance aircraft design is to design a high lift system within the given spatial constraints without having to increase the wing area to e.g. meet landing speed requirements or, vice versa allow to have a lower wing area, and therefore less wing weight and less drag.

Recently and especially in the UP-WING project, very high aspect ratio wings have been designed to increase the aerodynamic performance. In addition to a careful aerodynamics and structural design due to the high flexibility of a high aspect ratio wing, the wing tip chord will be very short and can be easily subject to stall, especially at take-off conditions. However, wing tip stall has to be avoided since often it is not symmetric causing roll instability and is in addition coupled with loss of aileron effectiveness that will be in the wake of the area of separated flow.

First starting from a baseline cruise wing configuration, and taking into account all design objective and constrains, the high-lift system has been designed. Nevertheless, taking into account the short wing tip chord, solutions to avoid wing tip stall are required. Multifunctional vortex generators have been selected as solutions to avoid wing tip stall. They will have a dual use, will be used as conventional vortex generators to avoid wing tip stall in low-speed conditions, but will also be used as load alleviation devices in high-speed conditions. For this reason, the Vortex generators will be equipped with actuators that will allow for the vortex generators to be deployed, rotated or retracted so that they can be activated in the optimal conditions for specific flight conditions, but can also be retracted to avoid drag increase in flight conditions where they are not needed. In parallel also flow separation sensors will be developed so that a better knowledge of local flow conditions will support a better control of the vortex generators. These technical challenges have been addressed within the WP2.4 of UP-WING project and will be presented in this paper.

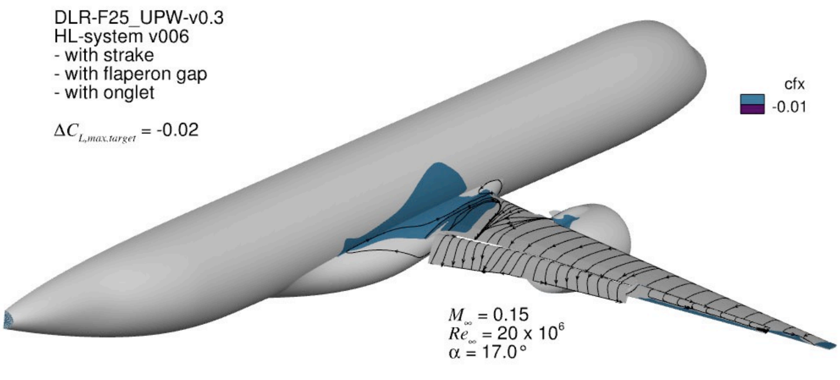


Figure 1: *The DLR-F25 configuration for the UP-WING project after iterations during the high-lift design.*

The high lift system was designed starting from an initial release of the generic research aircraft configuration DLR-F25 (Figure 1). The core requirement for the high-lift design was given by the Top-Level Aircraft Requirements (TLAR) of a high aspect-ratio wing aircraft to achieve an aircraft of ICAO approach category C [1].

The designed and delivered geometry achieves a maximum lift coefficient in landing configuration that exceeds the required values according to the TLAR by 1.5%. To avoid wing tip flow separation active multifunctional Vortex generators have been proposed in combination with flow separation sensors.

Acknowledgement

The project Ultra Performance Wing (UP Wing, project number: 101101974) is supported by the Clean Aviation Joint Undertaking and its members. The development of the DLR-F25 configuration was funded by the German Federal Ministry for Economic Affairs and Climate Action (BMWK) as part of the LuFo VI-2 project VIRENFREI (funding reference: 20X2106B).

References

[1] International Civil Aviation Organization, "Aircraft Operations" in *Procedures for Air Navigation Services*, vol. II, 2006.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-5
UP Wing Gust Load Alleviation Wind Tunnel Experiment: Overview on Controller Design Activities – Part 1

Thiemo Kier[1]*, Felix Stalla[1]*, Huub Timmermans[2], Andres Jurisson[2], Pierre Vuillemin[3]

[1] German Aerospace Center (DLR), Institute of System Dynamics and Control, Wessling, Germany
[2] Netherlands Aerospace Center (NLR), Anthony Fokkerweg 2, Amsterdam, Netherlands
[3] ONERA - The French Aerospace Lab, Toulouse, France
E-mail: thiemo.kier@dlr.de

Key Words: *Aircraft load control, wind tunnel experiment, numerical model, simulation, controller design, finite element method, doublet lattice method*

The UP Wing project investigates wing technologies to be used on a next generation short and medium range transport aircraft. One promising technique to further reduce structural weight is active load alleviation. To mature these load control functions, a wind tunnel experiment is conducted in transonic conditions within UP Wing WP2.3. Based on a reference configuration, a wing wind tunnel model is developed, featuring sweep and high aspect ratio. In the experiment, a gust generator introduces disturbances in the airflow, which then interact with the flexible wing, exciting its dynamics. The motion is sensed by distributed acceleration sensors, load estimators, and a (virtual) LIDAR. These measurements are provided as feedback / feedforward signals to the control algorithms to be designed. Three trailing edge control surfaces are actuated to reduce the loads resulting from the gust encounter.

These two presentations (Part. 1 and Part. 2) aim to give an overview on the activities underway to design gust load alleviation controllers to be tested in the wind tunnel. Part. 1 will focus on the numerical model used for model-based controller design, while Part. 2 will be concerned with the controller design itself. Challenges due to transonic flow conditions will be discussed and first results will be shown.

The first step in model-based controller design is to generate a numerical model to mathematically describe the wind tunnel experiment. Two types of models are created:

- a nonlinear simulation model for controller validation,
- a linear state-space model for controller synthesis.

The model includes a structural model derived from a finite element formulation, being condensed to a set of structural grid point. The aerodynamic model is based on the doublet lattice method. The aero-structural coupling employs a radial basis function approach. Control surfaces are modelled as a second order element, while the sensors are modelled purely geometrically, neglecting the very fast sensor dynamics.

The challenge in controller design for transonic flow conditions is to accurately capture the nonlinear aerodynamic effects. The aerodynamic model is typically based on fast aerodynamic methods, such as the Doublet Lattice Method (DLM) based on linear potential theory are used to obtain frequency domain data for given reduced frequencies. For transonic conditions this is not sufficient, hence the DLM matrices will be corrected with data obtained from unsteady higher fidelity CFD simulations.

For gust modeling the Loewner framework will be used. This framework poses an alternative approach to transferring frequency domain gust aerodynamics to the time domain, as the classical rational function approximation (e.g. Roger) is not well suited. The Loewner framework shows promising results in obtaining a low order gust model.

Any additional sources of detrimental nonlinearities, e.g. free-play, actuator rate limits, etc. also need to be addressed in the modelling process to allow for an meaningful assessment of the controller performance.

References

[1] Kier, T., and Looye, G., "Unifying Manoeuvre and Gust Loads Analysis Models," International Forum on Aeroelasticity and Structural Dynamics (IFASD), Seattle, USA, 2009. URL https://elib.dlr.de/97798/.

[3] Kier, T. M., and Hofstee, J., “VarLoads - eine Simulationsumgebung zur Lastenberechnung eines voll flexiblen, freifliegenden Flugzeugs,” Deutscher Luft- und Raumfahrtkongress (DLRK), DGLR, Dresden, Germany, 2004.
[9] Albano, E., and Rodden, W. P., “A Doublet-Lattice Method for Calculating Lift Distributions on Oscillating Surfaces in Subsonic Flows,” AIAA Journal, Vol. 7, No. 2, 1969. https://doi.org/10.2514/3.5086.
[14] Beckert, A., and Wendland, H., “Multivariate Interpolation for Fluid-Structure-Interaction Problems Using Radial Basis Functions,” Aerospace Science and Technology, Vol. 5, No. 2, 2001. https://doi.org/10.1016/S1270-9638(00)01087-7.
[15] Roger, K. L., “Airplane Math Modeling Methods For Active Control Design,” AGARD Conference Proceedings 228, Advisory Group For Aerospace Research and Development of NATO (AGARD), 1977.
[17] Tang, M., Böswald, M., Govers, Y., and Pusch, M., “Identification and Assessment of a Nonlinear Dynamic Actuator Model for Controlling an Experimental Flexible Wing,” CEAS Aeronautical Journal, Vol. 12, 2021. https://doi.org/10.1007/s13272-021- 00504-y.
[19] Quero, D., Kaiser, C., Vuillemin, P., and Poussot-Vassal, C., “A State-Space Model for Loads Analysis Based on Tangential Interpolation,” International Forum on Aeroelasticity and Structural Dynamics (IFASD), Savannah, USA, 2019. URL https://elib.dlr.de/128079/.
[22] Mayo, A., and Antoulas, A. C., “A Framework for the Solution of the Generalized Realization Problem,” Linear Algebra and its Applications, Vol. 425, 2007. https://doi.org/10.1016/j.laa.2007.03.008.
[23] Fournier, H. et. al., “Robust Gust Load Alleviation of Flexible Aircraft Equipped with Lidar,” Journal of Guidance, Control, and Dynamics, Vol. 45, No. 1, 2022. https://doi.org/10.2514/1.G006084.
[24] Ripepi, M.; Verveld, M. J.; Karcher, N. W.; Franz, T.; Abu-Zurayk, M.; Görtz, S. & Kier, T. M., “Reduced-order models for aerodynamic applications, loads and MDO“, CEAS Aeronautical Journal, Springer Nature, 2018, 9, 171-193
[24] Verveld, M. J. & Kier, T. M., „Reduced Order Model of Corrected Aerodynamic Influence Coeffcients for Aircraft Loads Analysis”, 58th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, 9-13 January 2017, Grapevine, TX, USA, 2017

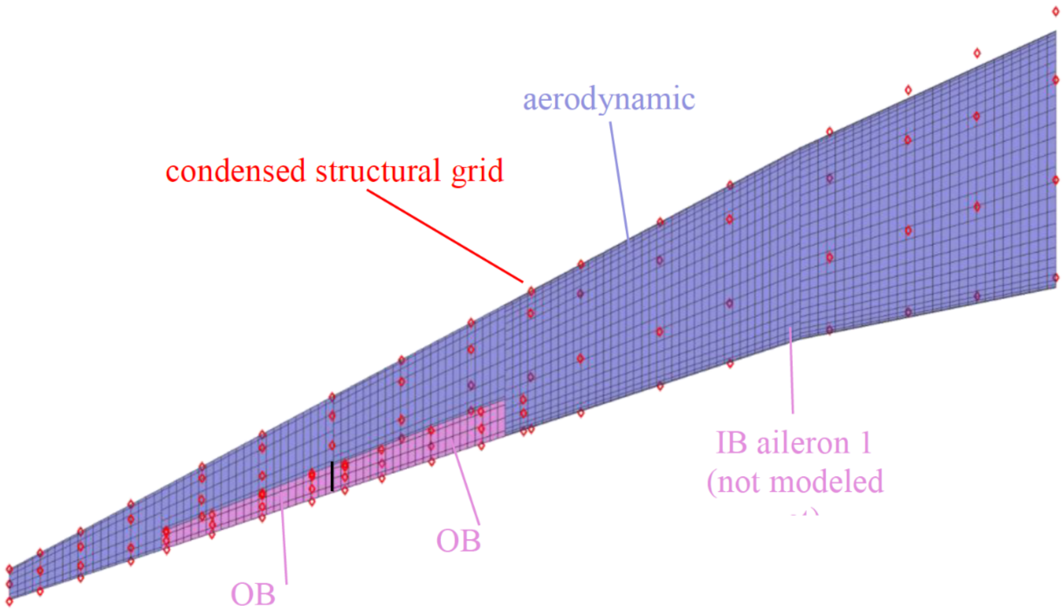


Figure 1: *Aerodynamic and structural model of the UP Wing wind tunnel model*

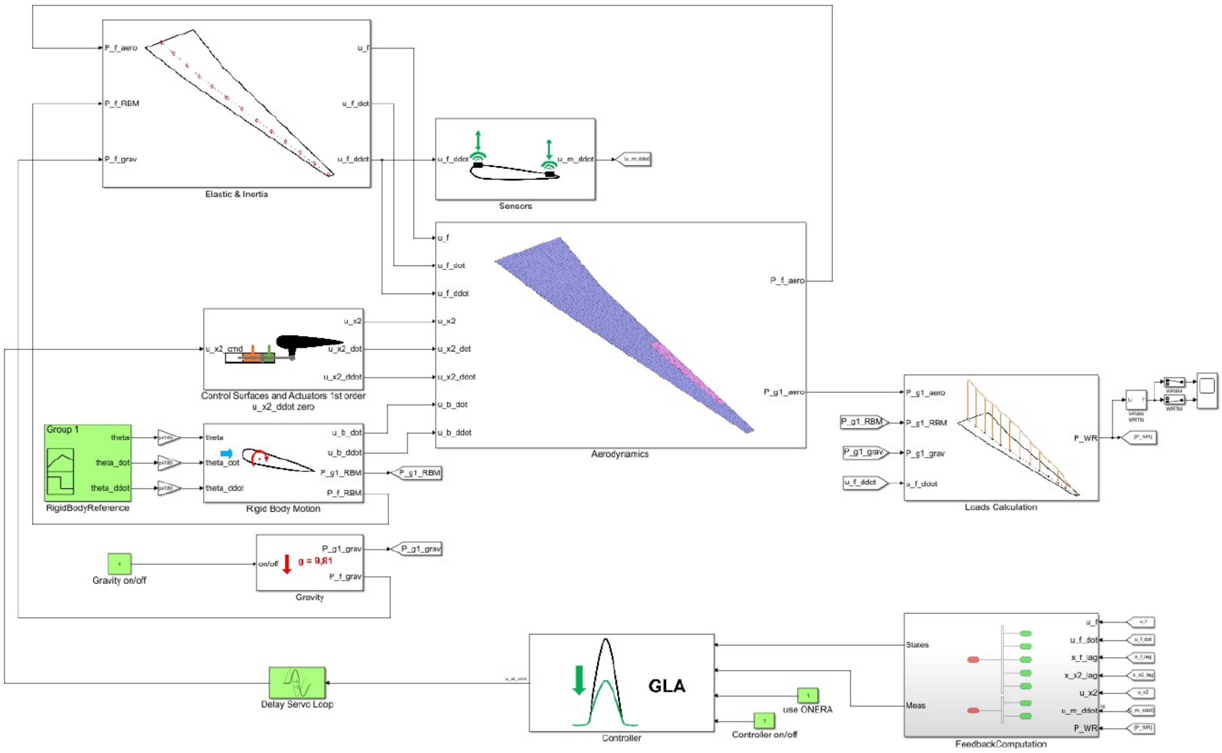


Figure 2: *Top level of the nonlinear MATLAB-Simulink model for the UP Wing wind tunnel experiment*

STS 271-6

UP Wing Gust Load Alleviation Wind Tunnel Experiment: Overview on Controller Design Activities – Part. 2

Felix Stalla[1]*, Thiemo Kier[1]*, Pierre Vuillemin[2], Xuerui Wang[3], Ralf Senger[4]
[1] German Aerospace Center (DLR), Institute of System Dynamics and Control, Wessling, Germany
[2] ONERA - The French Aerospace Lab, Toulouse, France
[3] Technical University Delft, Delft, The Netherlands
[4] Technical University Berlin, Berlin, Germany
E-mail: felix.stalla@dlr.de

Key Words: *Aircraft load control, wind tunnel experiment, MATLAB-Simulink, controller design, robust control, output feedback, INDI, time delay*

The UP Wing project investigates wing technologies to be used on a next generation short and medium range transport aircraft. One promising technique is active load alleviation. To mature load control techniques a wind tunnel experiment is conducted within UP Wing WP2.3. Based on a reference configuration, a wing wind tunnel model is developed, featuring sweep and high aspect ratio. In the experiment, a gust generator introduces disturbances in the airflow, which then interact with the flexible wing, exciting its dynamics. The motion is sensed by distributed acceleration sensors, load estimators, and a (virtual) LIDAR. These measurements are provided as feedback / feedforward signals to the control algorithms to be designed. Three trailing edge control surfaces allow to reduce the loads resulting from the gust encounter.

These two presentations (Part 1 and Part 2) aim to give an overview on the activities underway to design gust load alleviation controllers to be tested in the wind tunnel. Part 1 will focus on the numerical model used for model-based controller design, while Part 2 will be concerned with the controller design itself. Challenges due to transonic conditions will be discussed and first results will be shown.

The numerical model previously presented (Part 1) is used in model-based controller design. The controllers are synthesized using the linear state-space model of reduced order. The validation of the controllers is performed in the nonlinear MATLAB-Simulink environment. Aim of the gust load alleviation control is to reduce the integral loads at the wing-root, while ensuring local loads at more outboard spanwise locations are not unnecessarily increased.

Multiple control strategies will be examined by different partners. These include:

- Robust control using μ-synthesis and H-inf struct,
- Incremental nonlinear dynamic inversion (INDI),
- Output feedback with and without state estimation.

These control strategies will be explained in the presentation. Each control strategy is applied to multiple test scenarios. The baseline features distributed acceleration sensors as feedback variables. In a second case, the feedback from the wind tunnel balance can be interpreted as a load estimation. A third case uses feedforward signals as they would be supplied by a LIDAR system or angle of attack sensor.

An important feature in all controllers to be tested experimentally is sufficient robustness. This poses a great challenge in the UP Wing wind tunnel experiment, as the freestream velocity is similar to the one that a full aircraft would experience at cruising altitude, while the scale of the wing is approximately one-sixteenth of the full wing. This leaves very little time to react to a gust interacting with wing. Dead time in hardware must be reduced, and a focus in the design of the controllers lies on achieving a good trade-off between robustness and performance.

Preliminary investigations of load alleviation potential have been conducted, analysing the effect of dead time, limits in actuator capabilities, freestream velocity, and gust parameters. Figure 1 shows the achievable load reduction when applying full state feedback LQR (LQG) and an aileron deflection limit of 10°, using the two outer ailerons only. The figure shows promising achievements, being able to reduce the peak-to-peak wing-root bending moment by about 40%. This number will naturally be significantly reduced when taking further constraints into account, e.g. no full state feedback being available, but it gives a good estimate of the maximum achievable reduction under optimal conditions.

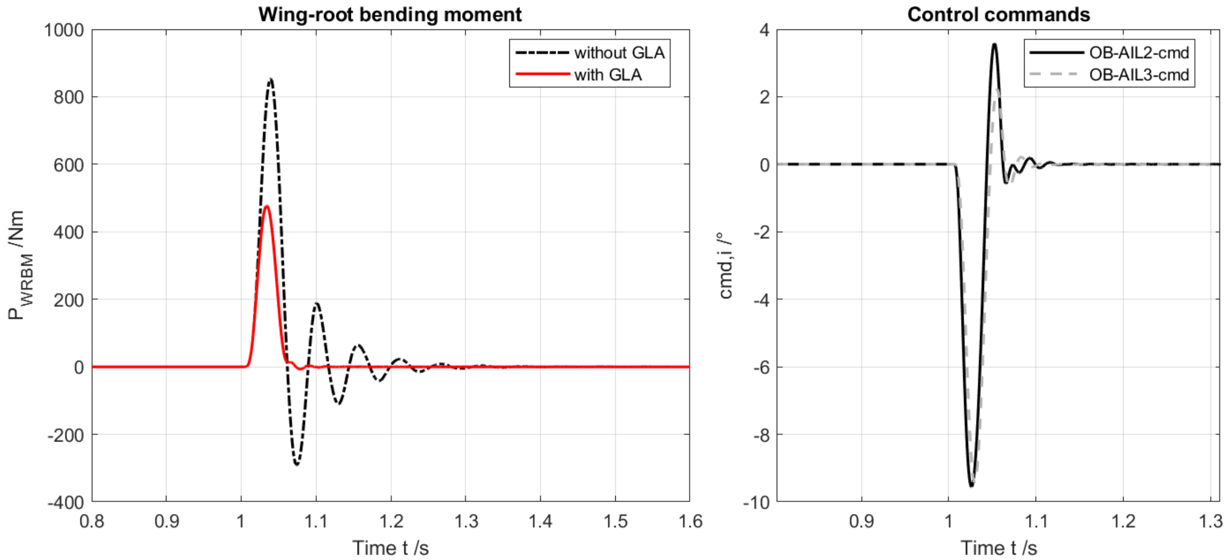


Figure 1: *Load reduction (left) and control surface deflection (right) using an LQR controller, 1-cos gust encounter*

References

[1] Skogestad, S., and Postlethwaite, I., Multivariable Feedback Control, 2nd ed., John Wiley & Sons, Chichester, UK, 2005.

[2] Stalla, F., Kier, T., Looye, G., and Pusch, M., "Aeroservoelastic Modeling and Robust Control for Gust Load Alleviation of an Experimental Wing", AIAA SciTech 2024 Forum, Orlando, USA, 2024. https://doi.org/10.2514/6.2024-1442.

[3] Wang, X., van Kampen, E., Chu, Q. P., and De Breuker, R., "Flexible Aircraft Gust Load Alleviation with Incremental Nonlinear Dynamic Inversion," Journal of Guidance, Control, and Dynamics, Vol. 42, No. 7, 2019. https://doi.org/10.2514/1.G003980.

[4] de Souza, A. d. R., Vuillemin, P., Poussot-Vassal, C., and Quero, D., "Gust Load Alleviation Using Reduced-Order Aeroelastic Models and Observer-Based Robust Control", Journal of Guidance, Control, and Dynamics, Vol. 46, No. 5, 2023. https://doi.org/10.2514/1.G007153.

[5] Silvestre, F. J., Guimarães Neto, A.B., Bertolin, R. M., da Silva, R. G. A., and Paglione, P., "Aircraft Control Based on Flexible Aircraft Dynamics",

STS 271-07

Multifunctional Active Vortex Generators

Pier Luigi Vitagliano[1], Bernardino Galasso[1,2], Francesco Capizzano[1,3], Pietro Catalano[1,4], Francesco D'Aniello[1,5], Donato de Rosa[1,6], Guiseppe Mingione[1,7,*], Carmelo Izzo[1,8], Antonio Concilio[1,9], Salvatore Ameduri[1,10], Luca Marino[11], Jurij Sodja[11,12]

[1] CIRA, via Maiorise, 81043, Capua, Italy, https://www.cira.it/en/, [1] p.vitagliano@cira.it, [2]b.galasso@cira.it, [3] f.capizzano@cira.it, [4] p.catalano@cira.it, [5] f.daniello@cira.it, [6]d.derosa@cira.it, [7] g.mingione@cira.it, [8] c.izzo@cira.it, [9] a.concilio@cira.it, [10]s.ameduri@cira.it

[11] TU Delft, Kluyverweg 1, 2629 HS Delft, The Netherlands, https://www.tudelft.nl/en/, [11] l.marino-1@tudelft.nl, [12] j.sodja@tudelft.nl

Key Words: *UP Wing, Clean-Aviation, high-lift vortex generators, load alleviation*

Abstract

Vortex generators (VGs) have been used for long time as a device to reduce flow separation. The concept of vortex generator is to energise the flow so that it will be more resistant to flow separation and therefore increase maximum lift and improve low speed performances. Vortex generators are usually fixed, therefore they are used in high lift, low speed conditions. In high speed they can cause an increase of drag. Therefore, the first idea is to develop active vortex generators so that they can be deployed only when needed avoiding the undesired drag increase in flight conditions when they are not necessary.

A non-conventional idea is to use vortex generators not only in low speed but also in high-speed conditions as a load alleviation device. During a manoeuvre, or in gust encounter, it could be required a reduction of wing aerodynamics forces to reduce wing design load and allow a design of a lighter wing. This work could be accomplished by vortex generators, that, if properly designed can be used to enhance flow separation, for example near wing tip, and therefore reduce wing loading. Vortex generators used for load alleviation should be usually larger and placed in specific wing position.

The objective of this work is therefore to design vortex generators with dual use, for both load alleviation and high lift improvements. This vortex generators should be active so to avoid drag increase in nominal cruise condition.

First numerical analysis using 2D and 3D CFD simulations was performed to define the design criteria for sizing and selection of the actuator and size of the vortex vane. Functionality tests will be performed in CIRA transonic wind tunnel. Test objective I to evaluate actuators effectiveness in actual load conditions.

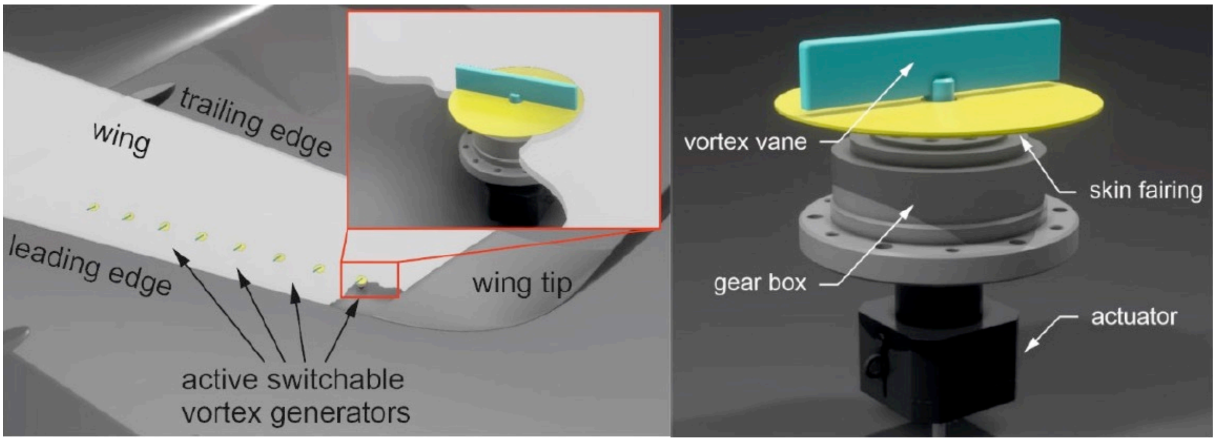


Fig. 1: *TUD Vortex Generator concept*

Vortex generators will be numerically modelled for the purpose of load alleviation and flow separation control. Two concepts for active vortex generators are proposed. The first concept, shown in Fig. 1, is a switchable vortex generator proposed by TU Delft which is equipped with an actuator to rotate the vortex vane in a desirable position with respect to the flow.

In the second concept, shown in Fig. 2, CIRA propose a deployable vortex generator featuring a movable part of the skin which is deployed by a shape memory alloy (SMA) actuator.

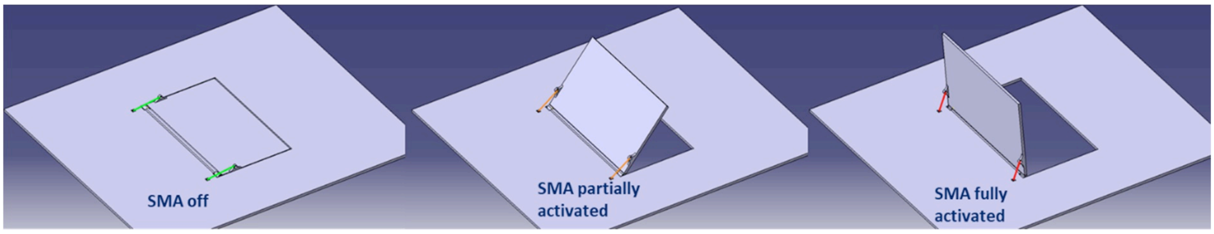


Fig. 2: *CIRA Vortex Generator concept*

The development of multifunctional active vortex generators will be presented. The proposed concept will be used for both stall prevention in low-speed conditions and load alleviation in high-speed conditions. In a first phase computational fluid dynamics is used to estimate geometric parameters of the vortex generators such as size, shape and position. The analysis is performed in steps: first 2D analysis, then 3D analysis on a wing section and finally analysis on the actual full 3D wing is performed.

The work is performed by CIRA and Delft University of Technology (TU Delft) in the framework of the Clean-Aviation Project UP Wing, in cooperation with Airbus, DLR and University of Bremen.

Acknowledgement

The project Ultra Performance Wing (UP Wing, project number: 101101974) is supported by the Clean Aviation Joint Undertaking and its members.

Further information

CORDIS: Ultra Performance Wing - UP Wing Project, https://cordis.europa.eu/project/id/101101974

YouTube: UP Wing - Ultra Performance Wing https://www.youtube.com/watch?v=MjrX6GvPGD8

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-8

Ultra-Performing Wing and Virtual Product Integration

Victoria Schmitz[1,*], Markus D. Krengel[1]

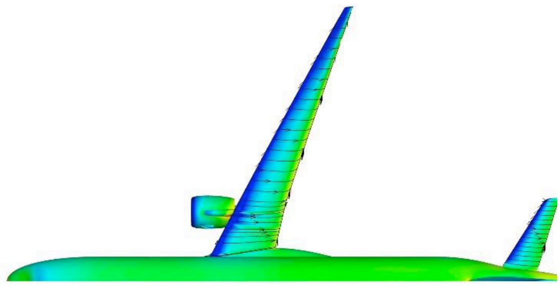
[1]German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Lilienthalplatz 7, 38108 Braunschweig, Germany, Victoria.Schmitz@dlr.de, markus.krengel@dlr.de, www.dlr.de

Key Words: *Virtual Product House (VPH), digital design, digital certification, ultra-performing wing, virtual product*

Short- and medium-range aircraft are responsible for a major part of the overall green-house gas emissions in aviation. To reduce this impact, the introduction of ultra-efficient, high aspect ratio (HAR) wing aircraft with improved aerodynamic performance and thus reduced fuel consumption can contribute to a more sustainable aviation. Since the integration of the moveables gets more complex on HAR and slender wings, detailed analysis of the wing and moveables structural and aerodynamic behaviour as well as of systems integration aspects are necessary.

The end-to-end simulation process developed within the DLR Virtual Product House (VPH) allows such multidisciplinary wing and moveables design tasks and the evaluation of optimum configurations [1]. In the European Clean Aviation project Ultra Performance Wing (UP Wing), the VPH simulation process utilizing low- and high-fidelity methods such as RANS based CFD-CSM coupling (DLR-Tau and MSC Nastran) [2] is used to perform an analysis of the DLR-F25 configuration with different wingspans. The span scaling of the wing planform is provided by a coupled overall aircraft design (OAD) process based on semiempirical methods. The detailed results from the VPH process are fed back into the OAD to evaluate the target function of a combined block fuel parameter for operationally relevant study missions. On the resulting trade curve of fuel efficiency over wing span, the optimal span and wing moveables configuration is identified.

a)



b)

Figure 1: *DLR-F25 baseline aircraft with 45m wingspan:*
a) *Skin friction distribution and stream traces on wing upper side;*
b) *Wing deformation for -1g and 2.5g*

The main focus here, however, is a discussion of the trade-curve results for selected span variations with specific focus on the aerodynamic effects.

References

[1] Lange, Fabian & Zakrzewski, Alexander et al. (2021). Digital Multi-Disciplinary Design Process for Moveables at Virtual Product House

[2] T. Wunderlich, S. Dähne, L. Reimer und A. Schuster, „Global Aero-Structural Design Optimization of Composite Wings with Active Maneuvre Load Alleviation“, Deutscher Luft- und Raumfahrtkongress 2020, Online, 2020

Acknowledgement:

The project Ultra Performance Wing (project number: 101101974) is supported by the Clean Aviation Joint Undertaking and its members.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-9

Enhanced Damage Tolerance Allowable for Improved Composite Wing Sizing

Tobias Wille[1,*], Raffael Bogenfeld[1], Andreas Schuster[1], Sascha Dähne[1] and Tobias F. Wunderlich[1]

[1]DLR, Lilienthalplatz 7, 38108 Braunschweig, Germany, tobias.wille@dlr.de, https://www.dlr.de

Key Words: *Structural Modelling, Composite Wing Sizing, Damage Tolerance*

Abstract

Aircraft design, particularly in the preliminary phase, requires damage tolerance (DT) considerations as a part of the structural optimization. As early design phases do not allow comprehensive test campaigns or detailed numerical analysis, the conventional DT assessment methods limit the design space and restrict the DT strain allowable. We present an innovative method that incorporates DT into the composite structural optimization process, aligning with guidelines for damage-tolerant aircraft design according to a no-growth approach. Focusing on the ultimate load static strength requirements for barely visible impact damage of the damage category 1, we have developed a DT allowable calculation method that is both practical and robust while enabling the identification of additional load carrying potential.

The core of our approach is a DT assessment process that systematically calculates the DT allowable through a series of iterative minimization procedures. This process is built upon established methods for analyzing impact, detectability, and residual strength. The innovative aspect of our approach lies in its ability to provide laminate-specific DT allowables by iterating over damage-prone interfaces in a layup, relevant thicknesses for each ply share configuration, and preselected ply share configurations. This methodology not only ensures a comprehensive coverage of the design space but also allows for the derivation of the most conservative DT allowable for the sizing process.

Our findings demonstrate a significant increase in the DT allowable – up to 50% – for optimized sublaminate configurations where 0°plies are avoided in the outermost layers of a laminate stack, [1]. When applied to the multidisciplinary aero-structural design of a composite wing, this results in a potential wing mass reduction of 5%, translating to a fuel consumption decrease of approximately 1.4%. These results highlight the potential of incorporating DT considerations in early design stages for more efficient and sustainable aircraft designs.

In addition to a more reliable DT prediction, the evaluation of uncertainty during the design is of great importance. Therefore, the resulting deterministic design process is extended by a probabilistic analysis. The presentation gives an overview of the particular inclusion of uncertainties by means of distributed material properties, as developed within the European UP Wing project.

Acknowledgement:

The new method for incorporating damage tolerance into the composite structural optimization process was developed within the DLR funded project Optimally Load-adaptive Aircraft (oLAF). Further developments on probabilistic analysis for considering uncertainties is developed within the project Ultra Performance Wing (UP Wing, project number: 101101974), supported by the Clean Aviation Joint Undertaking and its members.

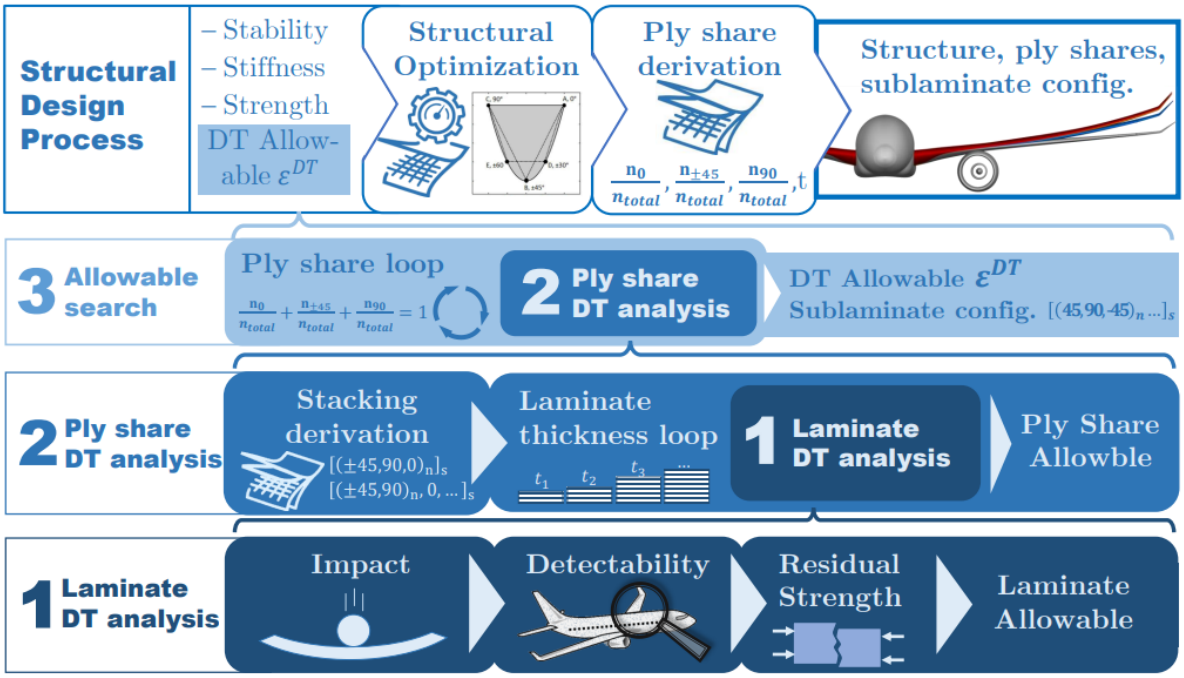


Figure 1: *Schematics of the Damage Tolerance assessment to calculate the allowable for the structural design process.*

References:

[1] R. Bogenfeld, S. Freund, S. Dähne, T. Wunderlich, T. Wille, Damage Tolerance Allowable Calculation for the Aircraft Design with Static Ultimate Load, Composite Structures. Elsevier. doi: 10.1016/j.compstruct.2023.117803. ISSN 0263-8223.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-10

Integration of Advanced Propulsion and Future Energy Carriers for Sustainable Aviation

Jens Friedrichs[1,2,*], Stanislav Karpuk[1,3]

[1] Cluster of Excellence SE²A-Sustainable and Energy-Efficient Aviation

[2] Institute of Jet Propulsion and Turbomachinery, Technische Universität Braunschweig, 38108 Braunschweig, Germany, j.friedrichs@ifas.tu-braunschweig.de

[3] Institute of Aircraft Design and Lightweight Structures, Technische Universität Braunschweig, 38108 Braunschweig, Germany, s.karpuk@tu-braunschweig.de

Key Words: *Overall aircraft design, Propulsion integration, Low-fidelity, High-fidelity, Reduced order model (ROM), Multidisciplinary optimisation*

Abstract

Global aviation as part of the megatrend "Mobility" has recovered from the Covid-19 crisis much faster than expected three years ago. This creates significant pressure on all measures to realize the challenging targets on environmental protection of at least an 80% reduction in greenhouse gas emissions as set by the IPCC to the climate neutrality goal defined by the European Union, both for 2050.

While other sectors, e.g. individual ground transportation do have options for reducing their carbon footprint e.g. through electrification, the very high power demands and energy densities in aviation do not allow for a direct transfer of such technologies. Liquid H2 (LH2) offers more energy density and on top to option of processing via gas turbines with their superior power density or fuel cells, which additional to the carbon footprint also fully eliminates any NOx emissions. On the other hand, large amounts of LH2 on-board an aircraft are driving the aircraft volume and thereby its drag. Sustainable aviation fuels (SAF) with drop-in properties are very advantageous from an operational perspective but (beside large-scale availability) do not offer the same climate impact reduction. Only synthetic kerosene on Power-to-liquid (PtL) basis using renewable electric power as a basis resource does offer a fully emission-neutral operation. From an energy cost perspective and assuming electric power from renewable resources as the general basis and at given cost level, the pure conversion efficiencies will drive the fundamental costs. For the Power-to-Gas (H2) process, liquid hydrogen is expected to be producible in large-scale processes at a level of roughly a factor of two compared to the electric power costs in 2050. For PtL e.g. via Fischer-Tropsch, the price increase will be by about 1.5, depending on process efficiency and up-scaling effects. These aspects lead to two basic principles:

I. Future aircraft have to be as energy efficient as possible since either the amount of energy to be stored on-board of a sustainable aircraft is limited or the costs per kWh of a sustainable fuel are significantly driving the tickets costs.
II. Integration of novel energy carriers and propulsion system does always require a full developed overall aircraft design (OAD) process which most probably will lead to different configurations being each an optimal answer to their combination of operating mission and propulsion system.

In turn, this will also lead to a future scenario, where one solution does not fit all, instead individual airframe concepts and propulsion integration solutions are required for individual forms following their distinctive function.

Since 2019 the Cluster of Excellence "Sustainable and Energy-Efficient Aviation – SE²A" is funded by the Deutsche Forschungsgemeinschaft (DFG) as part of the federal German excellence strategy. SE²A intends to establish the scientific and technological foundations for a sustainable future global air transport system and its required technologies. Two initial configurations have been conceptually designed: One with a backward-swept wing and one with a forward-swept wing (see Fig 1).

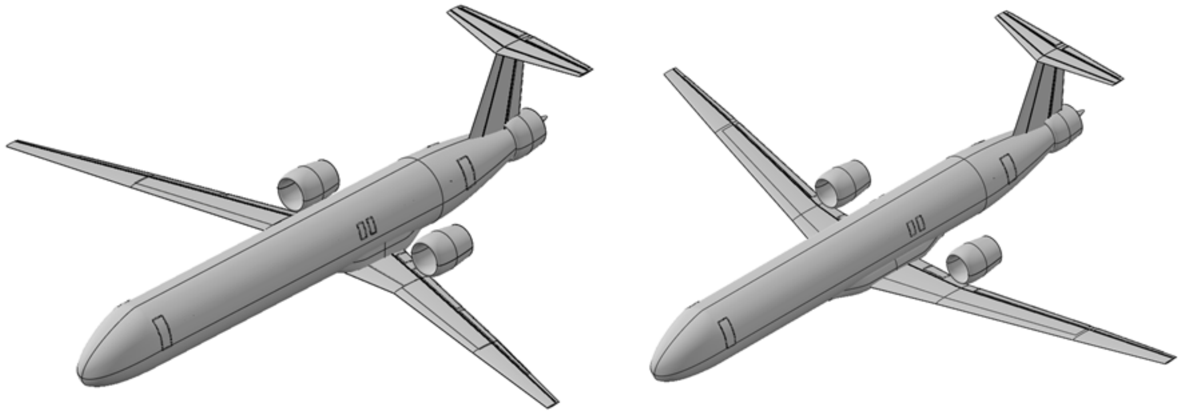


Fig. 1: *Starting configurations for mid-range reference aircraft design of SE2A Cluster*

References

[1] J. Friedrichs, A. Elham, C. Hühne, R. Radespiel and A. Bauknecht: Vehicle Technologies towards Sustainable and Energy Efficient. AIAA SciTech Forum. January 3-7, 2022, San Diego, CA. https://doi.org/10.2514/6.2022-0685.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 271-11

UNICADO – an Academic Conceptual Aircraft Design Environment for Research and Education

Eike Stumpf [1,*], Mirko Hornung[2], Andreas Bardenhagen[3], Volker Gollnick[4], Ingo Staack[5] and Andreas Strohmayer[6]

[1] RWTH Aachen University, ILR, Templergraben 55, 52056 Aachen, Germany, stumpf@ilr.rwth-aachen.de

[2] TU Munich, LLS, Boltzmannstr. 15/III, 85748 Garching near Munich, mirko.hornung@tum.de

[3] TU Berlin, ILR, Marchstr. 12-14, 10587 Berlin, Germany, andreas.bardenhagen@tu-berlin.de

[4] TU Hamburg, ILT, Blohmstr. 20, 21079 Hamburg, Germany volker.gollnick@tuhh.de

[5] TU Braunschweig, IFL, Hermann-Blenk-Straße 35, 38108 Braunschweig, Germany, ingo.staack@tu-braunschweig.de

[6] University Stuttgart, IFB, Pfaffenwaldring 31, 70569 Stuttgart, Germany strohmayer@IFB.Uni-Stuttgart.de

Key Words: *Aircraft Design, UNICADO, Academic Initiative*

Abstract

The paramount goal of aviation until 2050 is a substantial reduction of climate impact. Any amount of saved energy contributes directly to this. However, reducing the energy demand of air transport also helps closing the business case for costly alternative energy carriers, such as liquid hydrogen or sustainable aviation fuels. Due to these changed boundary conditions, it is reasonable to open and explore the full aircraft design space within research and development.

This requires specific skills and expertise and an approach different from current incremental development path. Thus, the future generation of engineers needs accordingly to be prepared and trained. This was the major trigger to start an academic initiative on national level in Germany, with all universities active in the field of aircraft design (RWTH Aachen, TU Berlin, TU Braunschweig, TU Hamburg, TU Munich, Univ. Stuttgart) joining the project UNICADO (UNIversity Conceptual Aircraft Design and Optimization) [1, 2].

The final paper will highlight two aspects:

1) UNICADO is intended to be continuously enhanced and maintained over the years and it will be published as open source under GNU General Public License in order to enable a broader community to contribute. Therefore, strict processes for modularization and (automatic) testing are applied.
2) UNICADO is introduced as design tool in aircraft design classes within the participating universities. In addition, specific aircraft design labs for concurrent design work will be established in each location. This allows to offer inter-university classes and student design competitions.

References

[1] F. Schültke and E. Stumpf, UNICADO: Aufbau und Etablierung einer universitären Flugzeugvorentwurfsumgebung. 70. Deutscher Luft- und Raumfahrtkongress, Bremen, 2021.

[2] M. Zimmnau, F. Schültke and E. Stumpf, UNICADO: Multidisciplinary analysis in conceptual aircraft design. CEAS Aeronautical Journal, 14 (1), 75-89, 2022.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 - 7 June 2024, Lisbon, Portugal

STS 272

Towards Next-Generation Aircraft Design with High-Fidelity Simulation Technologies

Chairs: Yoshiaki Abe[1], Keiichi Shirasu[2], Tomonaga Okabe[3], Shigeru Obayashi[4]

[1] Institute of Fluid Science, Tohoku University, Sendai, Japan,
yoshiaki.abe@tohoku.ac.jp, https://www.ifs.tohoku.ac.jp/eng/

[2] Department of Finemechanics, Tohoku University, Sendai, Japan,
keiichi.shirasu.c1@tohoku.ac.jp, https://www.mech.tohoku.ac.jp/en/

[3] Department of Aerospace Engineering, Tohoku University, Sendai, Japan,
tomonaga.okabe.a8@tohoku.ac.jp, https://www.mech.tohoku.ac.jp/en/

[4] Institute of Fluid Science, Tohoku University, Sendai, Japan,
s.obayashi@tohoku.ac.jp, https://www.ifs.tohoku.ac.jp/eng/

Session Abstract

Key Words: *Aircraft design, multi-objective design optimization, multi-physics simulation, aeronautical applications, structural optimization*

The proposed STS will cover computational methodologies and their applications to aircraft design with particular interest in technologies available for next-generation aircraft such as a "greener" aviation to mitigate an environmental impact. Examples of the computational methodologies include multi-objective design optimization, multi-physics simulation including fluid-structure interaction analysis, (unsteady) aerodynamic simulation, and geometrically nonlinear and/or material nonlinear analysis of structures.

The application of these methodologies may include but are not limited to a reduction of the airframe weight by utilizing carbon-fiber-reinforced plastics (CFRPs) and/or thermoplastics (CFRTPs), optimization of the stacking sequence of CFRPs, a reduction of the aerodynamic drag by laminarization technology, and design constraints considering flutter and buffet boundaries by fluid-structure interaction analyses. Furthermore, presentations from industries and governmental organizations are also welcome to introduce a current status and potential problems on the development of aircraft with existent simulation technologies.

Finally, we will not limit our scope to the examples above and encourage participants to propose a wide range of possible simulation technologies and frameworks that would benefit the development of a new-generation aircraft.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 272-1

Influences of Multiple Winglets on the Aerodynamic Performance and Flow Field of a Civil Aircraft

Erina Kobayashi[1]*, Kazuhisa Chiba[1,2] and Masahiro Kanazaki[3]

[1] Graduate School of Informatics and Engineering, The University of Electro-Communications, 1-5-1, Chofugaoka, Chofu, Tokyo 182-8585, Japan;
[1] k2232054@edu.cc.uec.ac.jp, [2] kazchiba@uec.ac.jp

[3] Graduate School of Systems Design, Tokyo Metropolitan University, 6-6, Asahigaoka, Hino, Tokyo 191-0065, Japan; kana@tmu.ac.jp

Key Words: *Computational fluid dynamics, Aircraft, Multiple winglets, Aerodynamic characteristics*

In this study, we investigate the impact on the aerodynamic performance and the flow field of civil aircraft with multiple winglets through numerical fluid analysis. The concept of multi-winglets, inspired by the primary feathers of birds, aims to emulate their function in reducing induced drag by dispersing wingtip vortices [1]. Previous research [2] applied multi-winglets on an unmanned aerial vehicle and found that they had a similar effect on their functions in birds. An additional achievement was an increase in lift by narrowing the gap between winglets. However, there have been few studies of applying three or more winglets to civil aircraft, and the physical effects on the flow field resulting from the installation of multi-winglets remain unexplored.

Therefore, we installed three winglets on the NASA Common Research Model and conducted numerical analysis under take-off/ landing and cruise conditions. For the multi-winglet model, the conventional single blended winglet was divided into three sections spanwise, creating a configuration with different cant angles for each winglet. Additionally, we calculated for the single blended winglet before division. A comparison was made between the influences on the multi-winglet's aerodynamic performance and flow field.

As a result, under take-off/ landing conditions, the lift coefficient of the multi-winglet increased compared to the single winglet. This improvement is attributed to the acceleration of airflow between the winglets' slits, leading to a reduction in pressure on the upper surface of subsequent winglets. On the other hand, the drag coefficient increased under both take-off/ landing and cruise conditions compared to the single winglet. It is due to flow choking between the winglets' gaps, causing increased pressure drag as flow stagnates on the front surfaces of each winglet. Accordingly, it is necessary to adjust to each winglet's cross-sectional shape and cant angle, especially in conditions where drag reduction is crucial, such as take-off and cruising.

References

[1] V. A. Tucker. Drag reduction by wing tip slots in a gliding Harris' hawk, Parabuteo unicinctus. Journal of experimental biology, *198*(3):775-781, 1995.

[2] M. Lynch, B. Mandadzhiev and A. Wissa. Bioinspired wingtip devices: a pathway to improve aerodynamic performance during low Reynolds number flight. *Bioinspiration & biomimetics*, *13*(3), 2018.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 272-2

Identification of Losses from Engine-Airframe Interaction for a Passenger Aircraft through Integrated Simulation

Jan Mueller[1]*, Kazuhisa Chiba[1] and Yoshinori Oba[2]

[1] Graduate School of Informatics and Engineering, The University of Electro-Communications, 1-5-1, Chofugaoka, Chofu, Tokyo 182-8585, Japan, ma004016@edu.cc.uec.ac.jp, http://www.di.mi.uec.ac.jp

[2] Aero Engine, Space & Defense Business Area, IHI Corporation, 3975-18, Haijima, Akishima, Tokyo 196-8686, Japan, oba1132@ihi-g.com

Keywords: *Finite volume method, Sliding mesh interface, Frozen rotor, Unstructured mesh, Turbo engines, Fluid mechanics*

Modern passenger airliners are designed with optimal efficiency in mind. Therefore, aircraft are equipped with engines of ever increasing bypass ratio. However, the increased engine cross-section and the proximity between engine nacelle and wing cause several interaction phenomena that negatively impact aerodynamic efficiency. This is significant as airframe and engine are usually developed separately, without these interactions in mind.

We endeavour to identify these interaction phenomena through integrated simulations of airframe and running engine. The engine is modelled as a bypass channel with rotating fan based on NASA Rotor67 [1]. The airframe is based on the NASA Common Research Model (CRM). Airframe and rotor are discretized as two separate domains with unstructured tetrahedral meshes. Transition between the domains is realized through sliding mesh interfaces with the rotating domain treated as a frozen rotor.

Boundary conditions were selected to provide rotor inflow conditions similar to [1] to verify the rotor flow. Surface mesh quality improvements enable a more stable computation with improved convergence behaviour. Our results show how engine intake spill flow alters the pressure distribution at the wings, negatively impacting lift. The altered pressure distribution in turn affects the exhaust jet, which then impacts thrust.

References

[1] A.J. Strazisar, J.R. Wood, M.D. Hathaway and K.L. Suder, Laser Anemometer Measurements in a Transonic Axial-Flow Fan Rotor, *NASA TP-2879*, 1989.

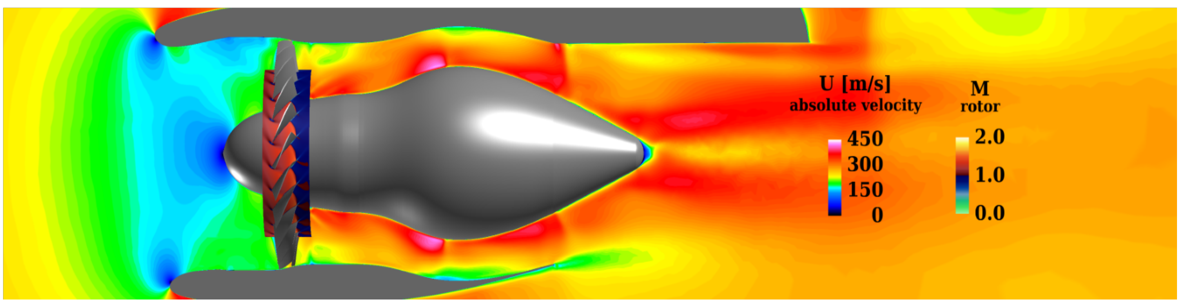


Figure 1: *Absolute flow velocity U in a vertical cut through the engine geometry and rotor Mach distribution M in a circumferential cut.*

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisbon, Portugal

STS 272-3

Upper Surface Morphing for Drag Minimization in NACA 64_1−612 Airfoil

P.N. Gautham Vigneswar[1]*, Shaikh Faruque Ali[2] and A. Arockiarajan[3]

[1,2,3] Department of Applied Mechanics and Biomedical Engineering, Indian Institute of Technology Madras, Chennai, Tamil Nadu, India,
[1] gautham.vigneswar@gmail.com, [2] sfali@iitm.ac.in, [3] aarajan@iitm.ac.in

Keywords: *Morphing wing, multi-objective shape optimization, drag reduction*

Research in morphing wing technology has shown immense promise in the journey towards green aviation [1]. The current study investigates the effects of upper surface profile on the aerodynamic performance of NACA 64_1 −612 airfoil with the aim of exploring its scope as an airfoil morphing technique. The study has been formulated as a multi-objective shape optimization problem by considering drag minimization and lift maximization as the two objective functions. Morphing of the upper surface has been implemented by adding a polynomial deflection function to its y coordinates and the aerodynamic coefficients are evaluated using XFOIL software. The parameters controlling the deflection function are considered as the design variables and the optimization problem is solved for angles of attack from 0° to 10° at a Reynolds number of 2.3×10^5 using the MATLAB® global optimization toolbox. This study provides a method to determine a theoretical upper limit for drag reduction and lift improvement that can be achieved using upper surface morphing. A maximum increase of around 17% and 77% have been achieved in lift coefficient (C_L) and lift-to-drag ratio (C_L/C_D) values respectively. The pareto fronts for angles of attack (AoA) 1° and 7° are shown in Figures 1(a) and 1(b) and the percentage increase in C_L/C_D values for angles of attack (AoA) from 0° to 10° are shown in Fig. 2.

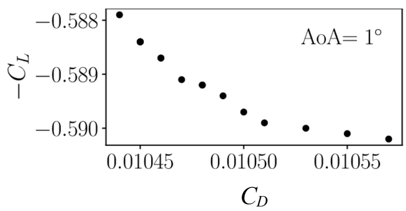


Figure 1(a)

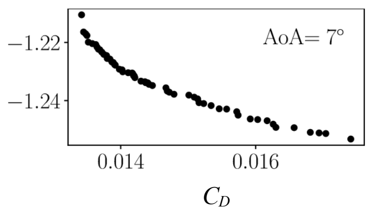


Figure 1(b)

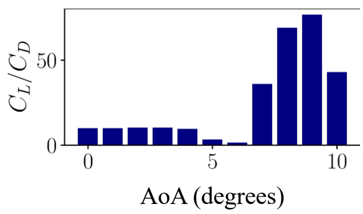


Figure 2

References

[1] Grigorie, T.L.; Botez, R.M. A Self–Tuning Intelligent Controller for a Smart Actuation Mechanism of a Morphing Wing Based on Shape Memory Alloys. Actuators 2023, 12, 350.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 272-4

Multi-objective Design of CFRP Composite Aircraft Wing with Next Generation Fibers and Resins

Yajun Liu[1]*, Tomoki Yamazaki [1, 2], Shugo Date[1,3], Toshio Nagashima[4] and Yoshiaki Abe[1,5]

[1] Institute of Fluid Science, Tohoku University, 2-1-1, Katahira, Aobaku, Sendai, 980-8577, Japan, [1]liu.yajun.d8@tohoku.ac.jp, [2] tomoki.yamazaki.s3@dc.tohoku.ac.jp, [3] shugo.date.s8@dc.tohoku.ac.jp, [5]yoshiaki.abe@tohoku.ac.jp

[4] Department of Engineering and Applied Sciences, Faculty of Science and Technology, Sophia University, 7-1, Kioicho, Chiyoda-ku, Tokyo 102-8554, Japan, nagashim@sophia.ac.jp

Key Words: *CFRP composite aircraft wing, multi-objective design, aerodynamic-structural simulation, NSGA-II optimizer*

The pursuit of carbon fiber reinforced plastic (CFRP) composite aircraft wings with high aspect-ratio planforms has garnered significant interest in recent research endeavors. In this study, we present a comprehensive multidisciplinary design framework that integrates a two-way coupled aerodynamic-structural simulation, multiscale CFRP composites modeling, structural sizing techniques, and a NSGA-II multi-objective genetic optimization algorithm. This innovative approach enables the design of CFRP composite aircraft wings with diverse planforms and airfoil shapes. The proposed multi-objective design framework incorporates the aerodynamic drag coefficient and wing structural weight as objective functions, with wing planform parameters and airfoil shapes serving as design variables. Furthermore, the study explores the impact of various fiber types and resin types, including both thermoset and thermoplastic resins, on the aircraft wing design. Through rigorous analysis, we aim to investigate the effects of next-generation fibers and resins on CFRP composite aircraft wings featuring different shapes. The study provides a significant step forward in optimizing CFRP composite aircraft wings, offering an understanding of the intricate relationship between material selection and aerodynamic performance.

References

[1] S Date, Y Abe, and T Okabe. Effects of fiber properties on aerodynamic performance and structural sizing of composite aircraft wings. Aerospace Science and Technology, 124 (2022): 107565.

[2] K Deb, A Pratap, S Agarwal, and T Meyarivan. A fast and elitist multiobjective genetic algorithm: NSGA-II. IEEE Trans. Evol. Comput., Vol. 6, No. 2, pp. 182–197, Apr. 2002.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3 – 7 June 2024, Lisbon, Portugal

STS 272-5

Structural Cohesive Element for the Modelling of Delamination between Thin Shells without Cohesive Zone Limit

Xiaopeng Ai[1], Boyang Chen[2,*] and Christos Kassapoglou[3]

Faculty of Aerospace Engineering, TU Delft, Kluyverweg 1, 2629 HS Delft, The Netherlands
[1] x.ai@tudelft.nl, https://www.tudelft.nl/staff/x.ai/
[2] b.chen-2@tudelft.nl, https://www.tudelft.nl/staff/b.chen-2/
[3] c.kassapoglou@tudelft.nl, https://www.tudelft.nl/staff/c.kassapoglou/

Key Words: *Composites; Cohesive zone modeling; delamination*

Cohesive element has been widely-used for the modelling of delamination between plies in a composite laminate. Despite its popularity, cohesive elements suffer from a well-known limit on the mesh density – the element size must be much smaller than the cohesive zone size. Otherwise, the steeply varying stress field in the cohesive zone cannot be integrated accurately, often leading to under-estimation of the peak cohesive stress and hence a delayed prediction of damage onset. This results in non-conservative strength predictions for delamination [1].

Based on the earlier work in 2D [2], this work develops a new set of elements for composite plies and their interfaces. The core insight is that the kinematics of the delaminated interface should follow from the geometries of the surfaces of the neighbouring plies which are slender shell-like structures. This means that the traditional approach of using solid-like cohesive elements (i.e., C0 continuous with polynomial shape functions) for delamination is inappropriate, as they are not designed to represent the kinematics of thin shell surfaces. Therefore, in this work, the plies will be modelled with C1-continous Kirchhoff-Love thin shell elements. A special cohesive element is formulated for the interface which conforms to the shell elements of the plies and shares their degrees of freedom. The proposed method is verified and validated on the classical benchmark problems of Mode I, Mode II and mixed-mode delamination. In addition, the Single-Leg Bending problem of a complex angle-ply laminate is modelled with the proposed method. All the results show that the mesh size with the new method can be several times larger than that allowed by the current limit on mesh density while retaining solution accuracy. This would then enable the accurate modelling of composite delamination without worrying about the current cohesive zone limit on mesh density, thereby saving a lot of computing time.

References

[1] A. Turon, C. G. Davila, P. P. Camanho and J. Costa. An Engineering solution for the mesh size effects in the simulation of delamination using cohesive elements. Engineering Fracture Mechanics, 74(10):1665-1682, 2007.

[2] R. Russo and B. Chen. Overcoming the cohesive zone limit in composites delamination: modeling with slender structural elements and higher-order adaptive integration. International Journal for Numerical Methods in Engineering, 121(24):5511–5545, 2020.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 272-6

Laminarization of Supersonic Three-dimensional Boundary Layer by Sinusoidal Roughness Elements

Makoto Hirota[1,*], Shoya Niwano[1], Yuki Ide[2], Yuji Hattori[1] and Shigeru Obayashi[1]

[1] Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi, 980-8577, Japan, makoto.hirota.d5@tohoku.ac.jp, obayashi@ifs.tohoku.ac.jp

[2] Aviation Technology Directorate, Japan Aerospace Exploration Agency, Chofu, Tokyo 182-8522, Japan

Keywords: *Supersonic boundary layer, Crossflow instability, Laminarization, Drag reducing roughness, Direct numerical simulation*

Reducing the frictional drag of aircraft is an important issue for realizing a supersonic passenger aircraft because its poor fuel efficiency is one of the major concerns. Due to large swept angle of main wing and high Reynolds and Mach numbers, boundary layer around the wing is three-dimensional and growth rate of crossflow instability tends to be very large at position close to the leading edge. This leads to laminar-to-turbulent transition upstream and significantly increases skin friction over the wing surface.

For a transonic three-dimensional boundary layer, a direct numerical analysis [1] shows that this crossflow-induced transition can be suppressed strongly by placing sinusoidal roughness elements (SREs) near the leading edge, if the shape parameters of SREs such as height and angle are chosen optimally. The induced distortion of mean flow profile contributes to stabilization of three-dimensional boundary layer, and thereby the transition position is shifted downstream. A wind tunnel experiment [2] was also conducted and successfully verified the expected laminarizing effect of SREs on a swept flat plate at 30m/s wind speed.

In this study, the same laminarizing method by SREs is numerically examined for supersonic three-dimensional boundary layers on flat plate. Because of the large growth rate of crossflow instability, it turns out to be difficult to suppress it by SREs for the case of large swept angle. Nevertheless, by improving the shape of SREs, the suppression effect and transition delay can be observed at swept angles 27 degrees and 46 degrees when the local Mach number near the leading edge is about 1.5. This result indicates the possibility of applying SREs on the supersonic aircraft's wing for drag reduction.

References

[1] M. Hirota, Y. Ide, Y. Hattori. Design Basis of Sinusoidal Roughness Elements for Enhanced Laminarizing Effect on Three-dimensional Boundary Layer, AIAA SciTech Forum 2024, AIAA-2024-0889, 2024.

[2] S. Suzuki, A. Yakeno, Y. Konishi, N. Tokugawa, M. Hirota, H. Takami, S. Obayashi. Experimental Validation of the Suppression of Crossflow Instability by Sinusoidal Roughness Elements, AIAA SciTech Forum 2024, AIAA-2024-0891, 2024.

The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3—7 June 2024, Lisbon, Portugal

STS 272-7

On the Extension of the CFD Solver CODA for Turbomachinery Applications

Graham Ashcroft[1]*, Alexander Bleh[1], Hakki Birliker[1] and Pascal Post[2]

[1] Institute of Propulsion Technology, German Aerospace Center (DLR), Linder Hoehe, 51147 Cologne, Germany, graham.ashcroft@dlr.de

[2] Airbus Operations GmbH, Airbus-Allee 1, 28199 Bremen, Germany, pascal.post@airbus.com

Key Words: *High Performance Computing, Boundary Conditions, Turbomachinery, Fluid Mechanics*

The optimal integration of propulsion systems in the next generation of aircraft is a fascinating and challenging multidisciplinary design problem. To fully realise the combined potential of next generation aircraft and engine technologies an integrated multiphysics design approach is key. In this context the German Aerospace Center (DLR), the French Aerospace Lab (ONERA) and Airbus are currently developing CODA (**C**FD **O**NERA **D**LR **A**irbus), a next-generation CFD solver for aircraft and turbomachinery design, devised to be able to fully exploit current and future HPC architectures [1].

To efficiently simulate turbomachinery components a number of specific boundary conditions are necessary. In this paper we discuss the implementation of the so-called mixing-plane boundary condition in CODA. Due to the relative motion between adjacent blade rows in compressor or turbine stages, the flows within turbomachinery components are inherently unsteady. To nevertheless efficiently simulate such flows in the context of steady simulations, approximate artificial boundary conditions (mixing-planes), in combination with some form of non-reflecting boundary condition, are commonly used between blade rows [2]. The implementation of such mixing-plane boundary conditions (which are nonlocal in space) in an highly optimized HPC environment, such as CODA, is nontrivial. In this work we describe the mixing-plane boundary condition implementation in detail and outline the approach adopted to minimize its impact on code performance and scalability.

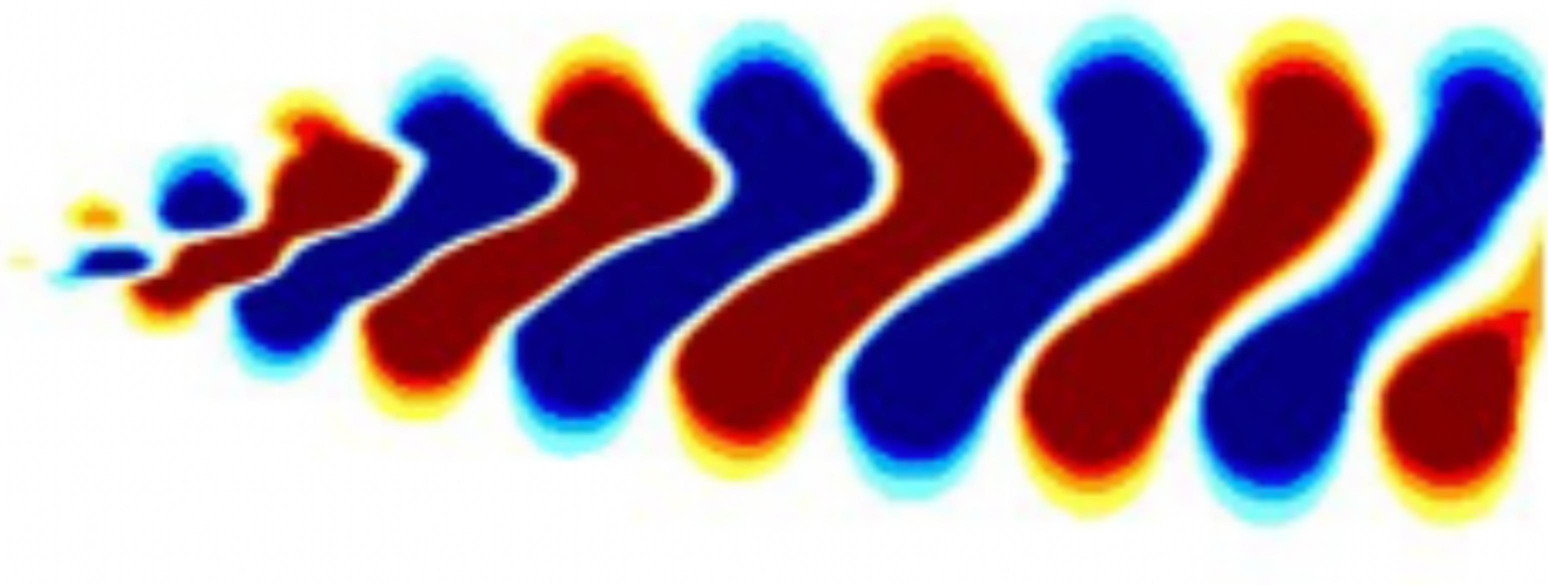
References

[1] S. Görtz, T. Leicht, V. Couaillier, M. Méheut, P. Larrieu and S. Champagneux, "CODA: A European Perspective for a Next-Generation CFD, Analysis and Design Platform", NATO AVT-366 Workshop on Use of Computational Fluid Dynamics and Analysis: Bridging the Gap Between Industry and Developers, 16-19 May 2022.

[2] M. Giles, 1991, "UNSFLO: A Numerical Method for Unsteady Inviscid Flow in Turbomachinery," Tech. Rep., Gas Turbine Laboratory Report GTL 205, MIT Dept. of Aero. and Astro.

Back Cover: STS247 *Prospects for Green Aircraft: Critical Technologies and Operational Aspects driving Overall Design*

STS247-5 Reduced Order Modelling for aerodynamic turbulent flows, By: Abderahmane MAROUF, Rajaa El Akoury, Braza Marianna, and Yannick Hoarau



ISBN: 978-84-127483-6-9