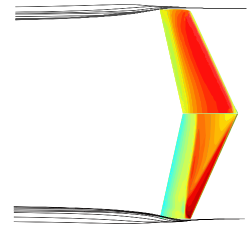
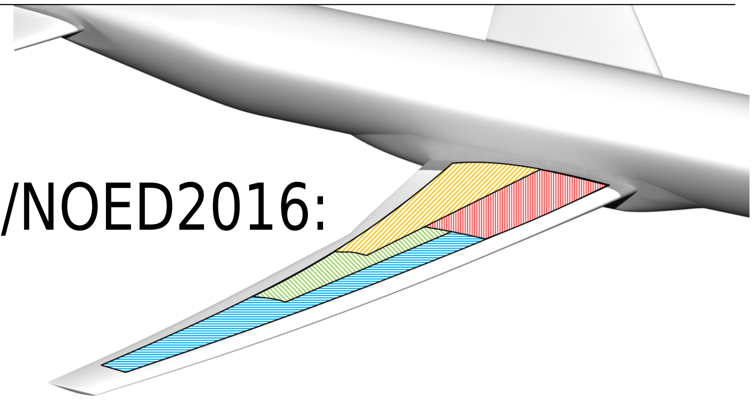
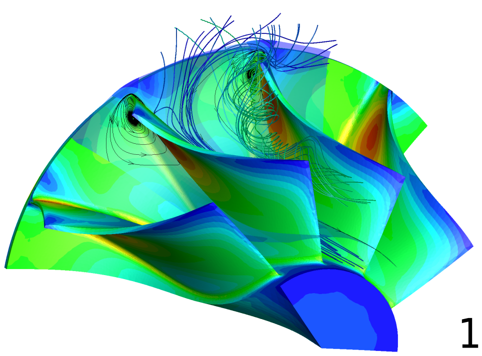
ASMO-UK
Technical University of Munich
University of Leeds
Queen Mary University of London

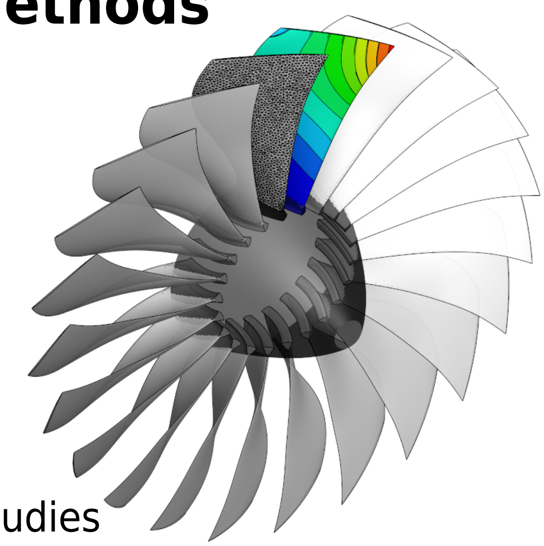




NOED 2016
11th ASMO UK/ISSMO/NOED2016:

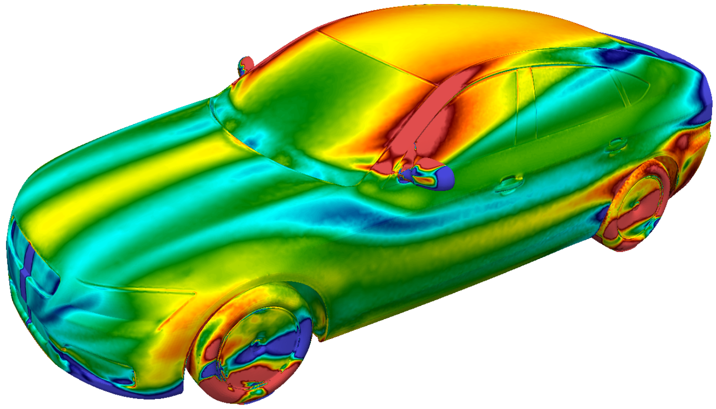
International Conference on Numerical Optimisation Methods for Engineering Design

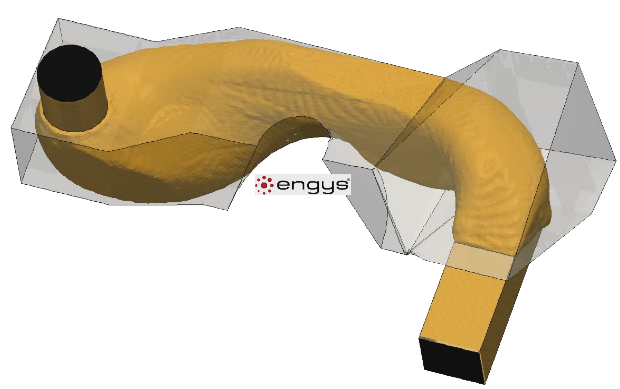




18-20 July 2016

Institute of Advances Studies
Techincal University of Munich





ASMO-UK/ISMMO
Technical University of Munich
University of Leeds
Queen Mary University of London

11th ASMO UK/ISSMO/NOED2016:

International Conference on Numerical Optimisation Methods for Engineering Design

18-20 July 2016

Institute of Advanced Studies
Technical University of Munich

Contents

1 Welcome

1.1 Conference Chairs

The conference is chaired by

Dr. Jens-Dominik Müller (Queen Mary University of London, UK)
Prof. Harvey Thompson (University of Leeds, UK)
Prof. Fabian Duddeck (Technische Universität München)

Prof. Thompson coordinates the EC project Amedeo[1], Dr. Müller coordinates the EC projects About Flow, IODA and Maddog[2]. It is the good level of funding by the EC under FP7 and H2020 that makes this conference possible, as well as a lot of the work presented here.

1.2 Scientific Committee

The scientific committee comprises a wide expertise in numerical optimisation methods for engineering design:

Prof. Fabian Duddeck (Technische Universität München, DE)
Dr. Laurent Hascoët (INRIA, Sophia Antipolis, FR)
Dr. Rob Hewson, Imperial College, London, UK)
Dr. Marcus Meyer (Rolls Royce, Dahlewitz, DE)
Dr. Jens-Dominik Müller (Queen Mary University of London, UK)
Prof. Uwe Naumann (RWTH Aachen University, DE)
Dr. Carsten Othmer (Volkswagen AG, Wolfsburg, DE)
Prof. Harvey Thompson (University of Leeds, UK)
Prof. Vassili Toropov (Queen Mary University of London, UK)
Prof. Tom Verstraete (Queen Mary University of London, UK)

Their contributions in shaping the call for papers and reviewing the submitted work is gratefully acknowledged.

We would also like to thank Juliet Jopson at Leeds University, Susan Barker and Vesna Milanovic at Queen Mary University of London, as well as Daniel Baumgärtner at TU Munich for their organisational support.

Dr. Müller would also like to thank his team of PhD students Siamak Akbarzadeh, Mateusz Gugala, Jan Hückelheim, Rejish Jesudasan, Pavanakumar Mohanamuraly and Orest Mykhaskiv. for their help with editing and proofreading webcontent and this book of abstracts.

[1] http://amedeo-itn.eu
[2] http:/{aboutflow,ioda,maddog}.sems.qmul.ac.uk

1.3 The Technical University of Munich

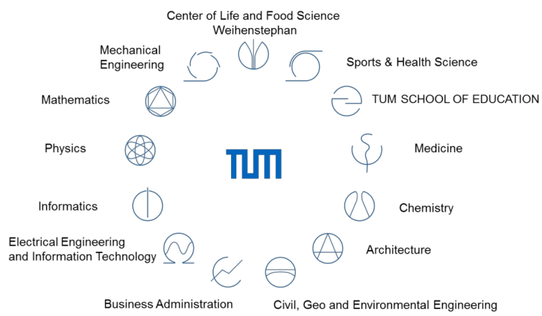
TUM is one of Europe's top universities. It is committed to excellence in research and teaching, interdisciplinary education and the active promotion of promising young scientists. The university also forges strong links with companies and scientific institutions across the world. TUM was one of the first universities in Germany to be named a University of Excellence. Moreover, TUM regularly ranks among the best European universities in international rankings.

The university was founded in 1868 by King Ludwig II to provide the state of Bavaria with a centre of learning dedicated to the natural sciences. The university played a vital role in Bavaria's transition from an agricultural to an industrial state âĂŞ and accelerated the pace of technological advancement across Europe.

TUM Campuses, from left to right: Central, Garching and Weihenstephan.

Today, TUM's 13 departments provide an excellent environment for research and for the education of 39,081 students. The university has a budget of € 1,258 million, which includes the university hospital. 8,911 graduates completed their degree in the academic year 2015. 3,024 of them were women, 1,549 international students. 1,021 doctorates were awarded by TUM in the same period. TUM top research clusters are focused on the following fields:

- Energy & Natural Resources
- Environment & Climate
- Health & Nutrition
- Mobility & Infrastructure
- Information & Communications



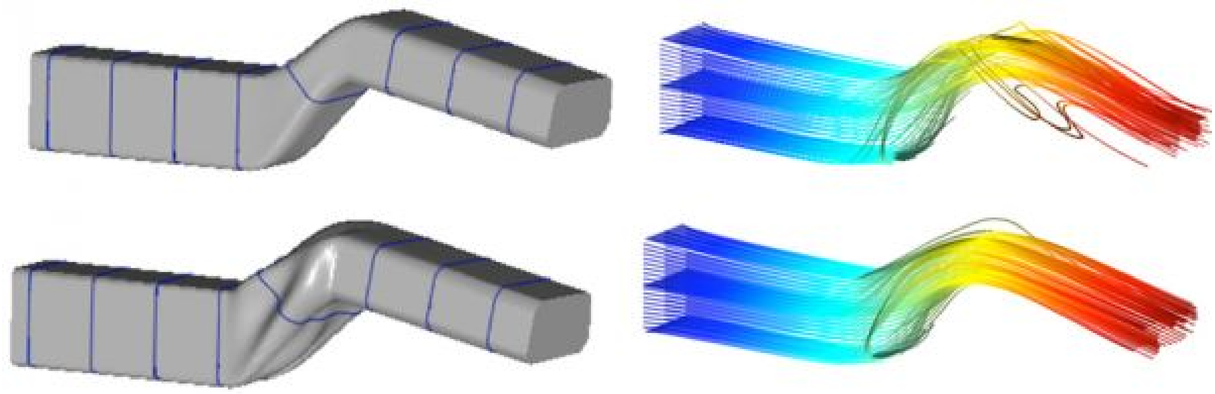
1.4 The Amedeo Initial Training Network

The future development of environment-friendly aircraft complying with recent European directives and research objectives (e.g. ACARE 2020 Vision; ACARE Beyond 2020 Vision) will be based on a systematic, model-based process where Multidisciplinary Design Optimisation (MDO) will be a key enabling technology. This has been clearly stated in Flightpath 2050: EuropeâĂŹs Vision for Aviation. The goal of MDO is to coordinate the individual disciplines affecting the design (e.g. aerodynamics, structural mechanics, acoustics,âĂę) toward a system design that is optimal as a whole, taking into account the key interactions between disciplines as well as competing objectives.

Over the last four years the AMEDEO (Aerospace Multidisciplinarity-Enabling DEsign Optimisation) Initial Training Network (http://www.amedeo-itn.eu/) has brought together academic and industrial experts from across Europe, including leading universities, research organisations, multinationals and innovative SMEs, to develop innovative MDO methods that can be used by Europe's aerospace industry to design new generations of energy-efficient aircraft. We are delighted that the 11th ASMO-UK/ISSMO/NOED2016 conference is providing us with an excellent opportunity to showcase our research achievements and highlight the future research challenges for the wider optimisation community. The AMEDEO team are looking forward to many interesting discussions with optimisation colleagues to stimulate future research collaborations and transfer knowledge and expertise across the wide range of industry sectors represented at the conference.

Professor Harvey Thompson, University of Leeds, UK, Coordinator of AMEDEO, July 2016.

1.5 The AboutFlow Initial Training Network



Adjoint-based methods have become the most interesting approach in CFD optimisation due to their low computational cost compared to other approaches. The development of adjoint solvers has seen significant research interest, and a number of EC projects on adjoint-based optimisation have been funded. In particular, partners of this proposal are members of the EC FP7 project FlowHead which has developed complete adjoint-based design methods for steady-state flows in automotive design.

The project addressed integration of the currently available shape and topology modification approaches with the gradient-based optimisation approach, in particular development of interfaces to return the optimised shape into CAD for further design and analysis, an aspect that currently requires manual interpretation by an expert user.

Most industrial flows have small levels of instability, which leads to a lack of robustness and instability of the adjoint, such as trailing edge vortex shedding in turbo-machinery. Many industrial applications are also partly unsteady such as bluff body separation in cars or fully unsteady such as vertical-axis wind turbines.

In unsteady adjoints 'checkpoints' of the flow solution at previous timesteps need to be recorded and algorithms for an effective balance between storage and recomputation need to be implemented. This involves significant memory and runtime overheads, efficient methods to overcome this are developed and implemented.

The methods developed in the project have been applied to realistic mid-size and large-scale industrial optimisation problems supplied by the industrial project partners ranging from turbo-machinery, to automotive to wind turbines.

About Flow is an Initial Training Network (ITN) funded by the European Commission running from November 2012 to October 2016. About Flow develops robust gradient-based optimisation methods using adjoint sensitivities for numerical optimisation of flows:

- robust adjoint solvers with full second order accuracy and broad range of modelling capability
- seamless integration of adjoint gradients into design chains
- steady-state approaches with robust convergence, application of unsteady adjoints in industrial design.

1.6 Practical information

1.6.1 Lecture theatres

The plenary lectures and the sessions on fluids and adjoint methods will be held in the main IAS Auditorium. The parallel sessions on structural and multi-disciplinary optimisation will be held in the Ludwig Prandtl Lecture Theatre, about 5 min walk. Signposts will guide you to this lecture theatre. The map in Fig. 1 shows the overall location, the three white 'U's in blue squares indicate the U-Bahn station you probably will arrive at.

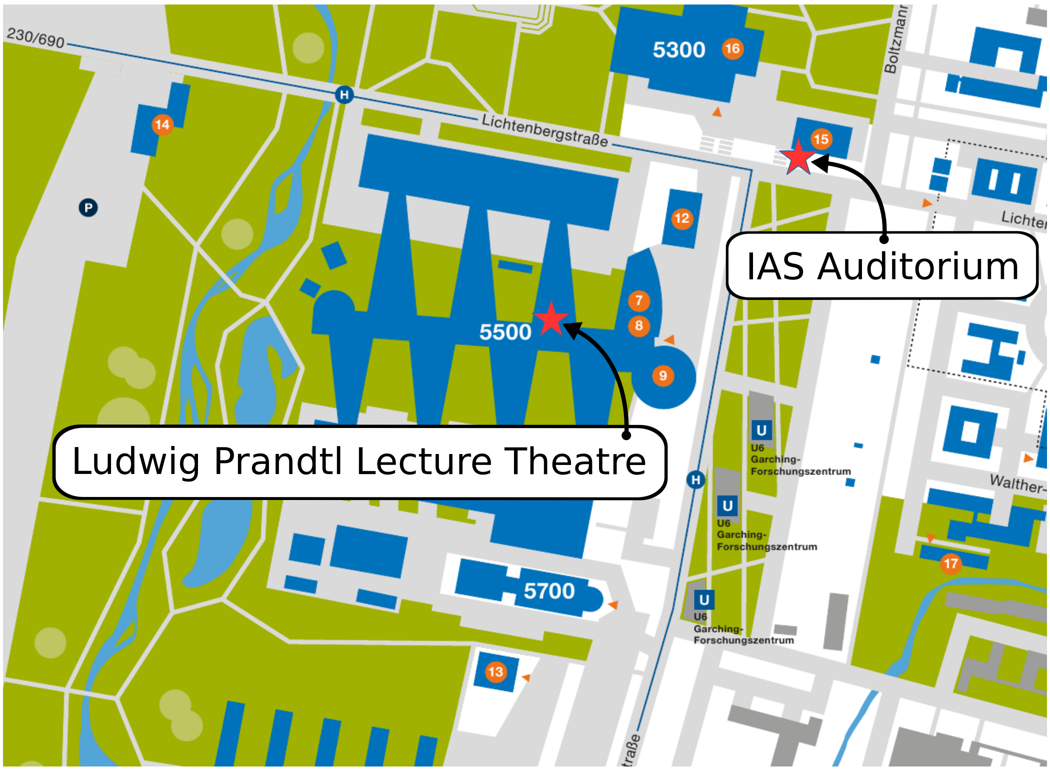


Figure 1: Location of the IAS Auditorium and the Ludwig Prandtl Lecture Theatre.

1.6.2 Coffee breaks and lunches

Coffee breaks and lunches will be served in the Faculty Club on the 4th floor. Please note that the space lobby is rented by the café in the lobby which is not involved in our catering.

Posters will be displayed in the Faculty Club and accessible during breaks.

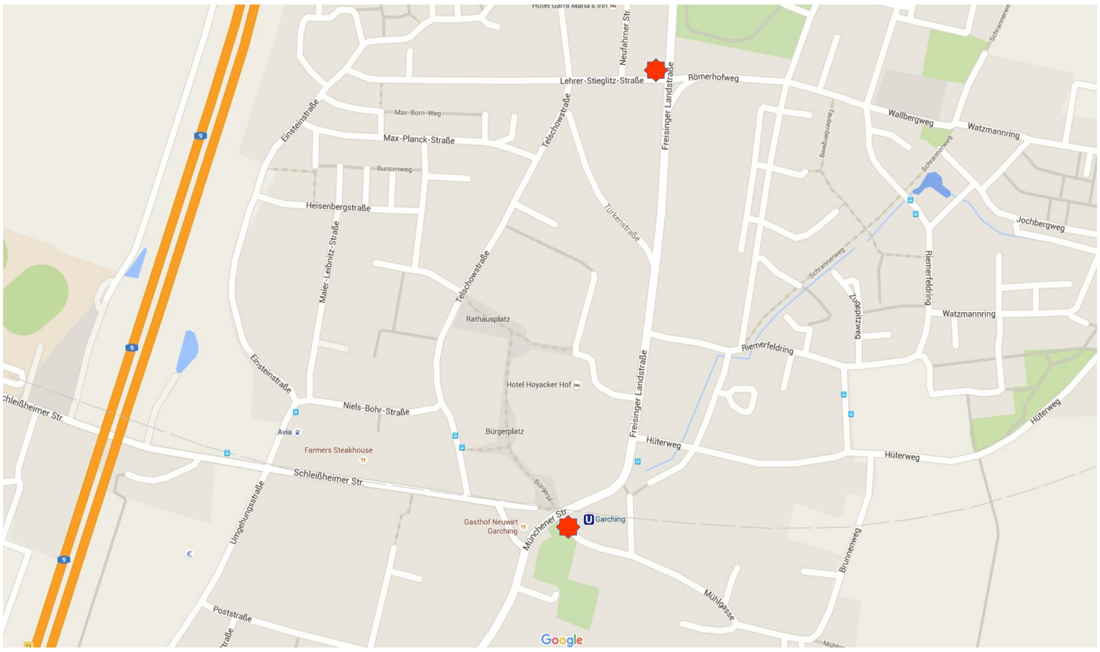
1.6.3 Conference dinner

The conference dinner will be held Tuesday evening at the Weihenstephan Brewery,

https://www.weihenstephaner.de

the oldest brewery in the world.

Buses will leave the IAS at around 18:20 to arrive for 19:00 dinner at the brewery. Buses will return after the dinner to the IAS and the hotels in Garching, see the plan in Fig 2.



Bus stop

Figure 2: Bus stops on return from the conference dinner.

1.6.4 Transport

Two S-Bahn train lines link to the airport, use

http://www.mvv-muenchen.de

and look up their travel planner which is available also in English.

Either of these two taxi firms shown below will be happy to take you to local destinations. Expect a fare of around € 35 between Garching and the airport, 25-30 min travel time in good traffic conditions.

Taxi Zentrale Freising (Freising, Garching and surroundings
Telefon: +49 8161 3666, +49 8161 19410

Taxizentrale München (Munich and surroundings)
Telefon: +49 89 21610 , +49 89 19410

2 Programme

2.1 Scientific programme: session overview

Monday 18 July 2016

8:00	Registration	
8:45	Welcome	
9:00	Fluid Mechanics and Heat Transfer optimisation	
10:20	Coffee break	
10:50	Aerospace Structural Optimisation Ludwig Prandtl Lecture Theatre	Adjoint solvers I IAS Auditorium
12:30	Lunch	
14:00	Invited Presentation: Eric Chaput (Airbus) "MDO for Highly Effective Aircraft Design"	
15:00	Turbomachinery Optimisation I	
15:40	Coffee break	
16:10	Structural Optimisation I Ludwig Prandtl Lecture Theatre	Adjoint solvers II IAS Auditorium
17:50	End of Monday session	

Tuesday 19 July 2016

9:00	Structural Optimisation I Ludwig Prandtl Lecture Theatre	Adjoint solvers III IAS Auditorium
10:20	Coffee break	
10:50	Keynote: U. Schramm (Altair) Design for Additive Manufacturing through Topology Optimization	
11:30	Design Parametrisation	
12:30	Lunch	
14:00	Invited Presentation: Christof Hinterberger (Faurecia) "From package space to final product design âĂŞ with adjoint CFD"	
15:00	Turbomachinery Optimisation II	
15:40	Coffee break	
16:10	Preliminary Aircraft Design Ludwig Prandtl Lecture Theatre	CFD Optimisation Benchmark Workshop: U-Bend IAS Auditorium
17:50	End of Tuesday session	
18:20	Buses leave from IAS for Weihenstephan	
19:00	Conference dinner at Weihenstephan brewery	

Wednesday 20 July 2016

9:00	Grid deformation and parametrisation
10:20	Coffee break
10:50	CFD Optimisation Benchmark Workshop: Drivaer, TUB Stator
12:30	Lunch
14:00	Invited Presentation: Joaquim Martins (Univ. Michigan) "MDO methodologies for Aerospace Industry"
15:00	Turbomachinery Optimisation III
15:40	End of conference

2.2 Sessions and speakers

2.2.1 Monday, 18 July

Monday 18 July 9:00-10:20:
Fluid Mechanics and Heat Transfer optimisation
IAS Auditorium

09:00 D. J. Munk, G.A. Vio, T. Kipouros, G.T. Parks
Computational Design for Micro Fluidic Devices using a Tightly Coupled Lattice Boltzmann and Level Set-Based Optimization Algorithm

09:20 J. Coddé, L. Vervecken
Topological Optimal Design and Experimental Validation of a Milled Liquid Cold Plate

09:40 A. Ranjan, S. Venkat, R. Maulik, F. Simon
Development of an accurate process workflow for simulation of a gas liquid cyclone separator in an open-source environment

10:00 Y. Caniou
Optimal design of a heat dissipating wheels for a 2CV race car

Monday 18 July 10:50-12:30:
Aerospace Structural Optimisation
Ludwig Prandtl Lecture Theatre

10:50 P. Lancelot, R. De Breuker
Aeroelastic tailoring for gust load alleviation

11:10 M.T. Bordogna, D. Bettebghor, R. De Breuker
Blending Constraints in Composite Wing Aeroelastic Optimization

11:30 K. Jovanov, R. De Breuker, M.M. Abdala, C. Blondeau
Multi-fidelity aeroelastic analysis and sensitivity analysis for gradient-based structural optimization

11:50 J. Ollar, R. Jones, V. Toropov
Incorporation of Bird Strike Requirements in MDO of an Aircraft Wing using Sub-space Metamodels

12:10 A. Arsenyeva, F. Duddeck
Wingbox adaptive parametric modeling and its application to structural optimization

Monday 18 July 10:50-12:30:
Adjoint solvers I
IAS Auditorium

10:50 S. Xu, S. Timme
Improved adjoint solver for complex flow conditions

11:10 T. Economon
Adjoint-based Optimization Techniques in SU2 with Applications to Industrial Flows

11:30 J.-D. Müller, M. Gugala, S. Xu, J. Hückelheim, P. Mohanamuraly, O. Imam-Lawal
Introducing STAMPS, an open-source discrete adjoint CFD solver using source-transformation AD

11:50 A. Sen, M. Towara, U. Naumann
Effective discrete adjoint OpenFOAM for volume and surface sensitivities

12:10 M. Gugala, J.-D. Müller
Output-based r-refinement using a flow-coupled system solve

Monday 18 July 14:00-15:00:
Invited presentation
IAS Auditorium

14:00 E. Chaput
MDO for Highly Effective Aircraft Design

Monday 18 July 15:00-15:40:
Turbomachinery Optimisation I
IAS Auditorium

15:00 R. Schlaps, J. Ollar, S. Shahpar, V. Toropov
Multi-disciplinary optimisation of a compressor rotor subjected to ice impact using metamodelling

15:20 J. Pohl, H.M. Thompson, V. Fico, G.A. Clayton
Turbine Stator Well Geometry Benefits âĂŞ Method Validation and Design Optimisation

Monday 18 July 16:10-17:30:
Structural Optimisation I
Ludwig Prandtl Lecture Theatre

16:10 H. Lu, M. Gilbert and A. Tyas
Use of layout optimization to identify optimal bracing systems in buildings

16:30 L. Huang and M. Mensinger
SOD: A Tool Auto-Generates the Preliminary Structural Design of Steel-Composite Office Buildings

16:50 S.K. Azad
Evaluating sensitivity of stochastic search techniques to profile list ordering in discrete sizing of steel frames

17:10 M. Bujny, N. Aulig, M. Olhofer, F. Duddeck
Evolutionary Crashworthiness Topology Optimization of Thin-Walled Structures

Monday 18 July 16:10-17:30:
Adjoint solvers II
Ludwig Prandtl Lecture Theatre

16:10 Y. Li, K. Claramunt, C. Hirsch
Automatic adjoint formulation with customized objective functions in an industrial CFD framework

16:30 Z. Dastouri, U. Naumann
Discrete Adjoint Approach for Commercial Computational Fluid Dynamics Codes using Algorithmic Differentiation Tools

16:50 A. Taftaf
Experiments on Checkpointing Adjoint MPI Programs

17:10 S. Akbarzadeh, J.-D. Müller, A. Sen, M. Towara, U. Naumann
Improved Fixed-Point Discrete Adjoint of simpleFoam

2.2.2 Tuesday, 19 July

Tuesday 19 July 9:00-10:20:
Structural Optimisation II
IAS Auditorium

09:00 J. Lüdeker, B. Kriegesmann
Probabilistic analysis of topologically optimized structures considering geometric imperfections

09:20 K. Kalnins, G. Jekabsons, E. Labans
Optimization for scaling up of plywood sandwich panels with rigid PU foam-cores

09:40 A. Ansola, A. Garaigordobil, E. VeguerÃa, O. Querin
Sequential element rejection and admission method (SERA) for topology optimization using a constraint on perimeter

Tuesday 19 July 9:00-10:20:
Adjoint solvers III
IAS Auditorium

09:00 J. Hückelheim, J.-D. Müller
Temporal and spatial checkpoint coarsening in adjoint shape optimisation for unsteady flow

09:20 M. Ghavami Nejad, E.M. Papoutsis Kiachagias, K. Giannakoglou
Aerodynamic Shape Optimization Using the Adjoint-based Truncated Newton Method

09:40 S.R. Islam
Adjoint-Based Numerical Error Estimation

10:00 M. Oriani, G. Pierrot
Advances in CFD Discretisation Schemes and Solution Algorithms for a Stable Discrete Adjoint

Tuesday 19 July 10:50-11:30:
Keynote lecture
IAS Auditorium

10:50 U. Schramm
Design for Additive Manufacturing through Topology Optimization

Tuesday 19 July 10:50-12:30:
Design parametrisation
IAS Auditorium

11:30 R. Jesudasan, M. Gugala, O. Mykhaskiv, J.-D. Müller, M. Banovic, S. Auriemma, A. Walther and H. Legrand
A comparison of node-based and CAD-based parametrisations in shape optimisation

11:50 D. Agarwal, C. Kapellos, T. T. Robinson and C. G. Armstrong
Parametric CAD model based shape optimization using adjoint functions

12:10 R. Najian Asl and K.-U. Bletzinger
A hybrid adjoint shape sensitivity analysis of fluid-structure interaction problems

Tuesday 19 July 14:00-15:00:
Invited presentation
IAS Auditorium

14:00 C. Hinterberger
From package space to final product design âĂŞ with adjoint CFD

Tuesday 19 July 15:00-15:40:
Turbomachinery Optimisation II
IAS Auditorium

15:00 M. Hassanine Aissa, C. Chahine, T. Verstraete
Surrogate-Model Assisted Evolutionary Optimization of an Axial Compressor Stator

15:20 T. Albring, S. Vitale, N. Gauger, M. Pini
Adjoint-Based Optimization of Multi-Stage Turbomachines using SU2

Tuesday 19 July 16:10-17:30:
Preliminary Aircraft Design
Ludwig Prandtl Lecture Theatre

16:10 A. Viti, T. Druot, A. Dumont, G. Carrier
Overall Aircraft Design Optimisation of a Nes Aircraft Configuration

16:30 D. Baumgärtner, K.-U. Bletzinger, A. Viti, A. Dumont
Node-based shape optimization in aircraft preliminary design

16:50 G. Serhat, T. Goncalves Faria, I. Basdogan
A Study on the Effect of Several Modelling and Analysis Parameters on the Optimization of Composite Laminates for Vibro-acoustic Requirements

17:10 C. Bach, R. Jebari, A. Viti, R. Hewson
A preliminary design method for optimizing composite forward-swept wings

Tuesday 19 July 16:10-17:30:
CFD Optimisation Benchmark Workshop: U-Bend
IAS Auditorium

16:10 G. Alessi, L. Koloszar, T. Verstraete, B. Blocken, J. van Beeck
Adjoint Shape Optimization of U-Bend Duct for Pressure Loss Reduction

16:30 L. Müller, T. Verstraete, J.-D. Müller
Adjoint based design optimization of a U-bend for minimized pressure losses

16:50 D. Kapsoulis, E.M. Papoutsis-Kiachagias, V. Asouti, K. Giannakoglou
U-Bend Optimization on the RBF4AERO Platform

17:10 O. Mykhaskiv, S. Auriemma, M. Banovic, J.-D. Müller, H. Legrand, A. Walther
Shape optimisation with differentiated CAD-kernel for U-bend testcase

2.2.3 Wednesday, 20 July

Wednesday 20 July 9:00-10:20:
Grid deformation and parametrisation
IAS Auditorium

09:00 G.K. Karpouzas, E. de Villiers
Geometric Immersed Boundaries (GIB): A New framework for applying boundary conditions in Finite Volume Method

09:20 G. Eleftheriou, G. Pierrot
Rigid Motion Mesh Morpher: a robust morphing tool for adjoint-based shape optimization

09:40 A.G. Liatsikouras, G.S. Eleftheriou, G. Pierrot, M. Megahed
CAD-Free Soft Handle Parameterization for Adjoint-Based Optimization Methods

10:00 P. Mohanamuraly, J.-D. Müller
A meshless optimised mesh-smoothing framework

Wednesday 20 July 9:00-10:20:
CFD optimisation Benchmark Workshop: Drivaer, TUB Stator
IAS Auditorium

10:50 N. Magoulas, M. Hartmann
Applications of the continuous adjoint method to car aeroacoustics

11:10 C. Vezyris, E.M. Papoutsis Kiachagias, K. Giannakoglou
Unsteady/Steady Continuous Adjoint Method Using a Block Coupled Solver in OpenFOAM: Application on the Drivaer Vehicle

11:30 I. Vasilopoulos, P. Flassig, M. Meyer, K. Reiche
Aerodynamic Optimization of the TurboLab Stator: A Comparative Study between Conventional and Adjoint-based Approaches

11:50 F. Gagliardi, K. T. Tsiakas, X. S. Tompoukis, K. Giannakoglou
Constrained Multi-Objective Optimization of the TU Berlin TurboLab Stator using Continuous Adjoint

12:10 S. Duckitt, C. Bisagni, S. Shahpar
Multiobjective Optimisation of a Compressor Stator using a 3D B-Spline Parametrisation

Wednesday 20 July 14:00-15:00:
Invited presentation
IAS Auditorium

14:00 J. Martins
MDO methodologies for Aerospace Industry

Wednesday 20 July 15:00-15:40:
Turbomachinery Optimisation III
IAS Auditorium

15:00 C. Chahine, T. Verstraete, L. He
Multidisciplinary Design Optimization of Aero-Engine Fan Blades

15:20 G. Ntanakas, M. Meyer
Generating Gradients for Turbomachinery Applications using the Discrete Adjoint Method

3 Abstracts

Abstracts are first given for invited and keynote papers, then for contributed papers which are ordered by session. An author index can be found at the back.

3.1 Invited and Keynote Presentations

Monday 18 July 14:00-15:00:

IAS Auditorium

Dr. Eric Chaput

Airbus, Flight Physics Capability Strategy

MDO for Highly Effective Aircraft Design

Dr. Eric Chaput joined Airbus in 1992 after a Ph.D. in Energetics and Optimisation, Post-doctoral positions in Experimental and Numerical Simulation at University of Poitiers, and six years' experience at Airbus Defence & Space working for ARIANE and HERMES programmes. He became subsequently CFD Research Manager, before managing Aerodynamics Methods and in 2004, Senior Manager of Flight Physics Methods. He is currently the leader of Airbus Flight-Physics capability strategy and a Senior Expert in Aerodynamics Flow Simulation Methods. He has long experience and interest in driving the development of Airbus engineering multidisciplinary capabilities, through collaborative research projects and industrialisation of integrated methods & tools within highly productive design environment.

Tuesday 19 July 14:00-15:00:

IAS Auditorium

Dr. Christof Hinterberger

Faurecia Emissions Control Technologies

From package space to final product design âĂŞ with adjoint CFD

Dr.-Ing. Christof Hinterberger is an expert for thermo-fluid analysis at the OEM automotive supplier Faurecia Emissions Control Technologies, in Augsburg Germany.

He graduated in Mechanical Engineering at the Technical University of Munich, and later completed his PhD in the field of Large Eddy Simulation at the Institute of Hydromechanics, University of Karlsruhe.

His current work at Faurecia focuses on developing CFD methods (e.g., for SCR-systems, simulation of soot loading and regeneration of diesel particulate filters, flow noise prediction of mufflers and tailpipes)

Over the past 10 years, his work has also included a strong focus on geometry optimisation using the continuous adjoint approach within OpenFOAM.

Abstract of the presentation

Package space restrictions and performance requirements (e.g., backpressure, uniformity of flow) present particular challenges for the layout of automotive exhaust systems. To address these difficulties, a geometry optimisation workflow has been developed at the automotive supplier Faurecia Emissions Control Technologies [1,2], which is based on the adjoint CFD method developed by Othmer et al. [3,4]. The workflow helps to find suitable solutions quickly during the product development process. The workflow will be presented with several examples.

[1] C. Hinterberger, M. Olesen: Automatic geometry optimization of exhaust systems based on sensitivities computed by a continuous adjoint CFD method in OpenFOAM. SAE 2010-01-1278.
[2] C. Hinterberger, M. Olesen, Industrial Application of Continuous Adjoint Flow Solvers for the Optimization Of Automotive Exhaust Systems, in proceedings of ECCOMAS CFD & Optimization conference, Antalya (2011)
[3] C. Othmer, E. de Villiers and H.G. Weller, Implementation of a continuous adjoint for topology optimization of ducted flows, AIAA-2007-3947.
[4] C. Othmer, A continuous adjoint formulation for the computation of topological and surface sensitivities of ducted flows, Int. J. Num. Meth. Fluids, Vol. 58, pp. 861-877, 2008.

Wednesday 20 July 14:00-15:00:

IAS Auditorium

Prof. Joaquim Martins

University of Michigan, Ann Arbor, Dept. of Aerospace Engineering

High-fidelity Adjoint-based Multidisciplinary Design Optimization of Aircraft Configurations

Joaquim R. R. A. Martins is a Professor at the University of Michigan, where he heads the Multidisciplinary Design Optimization Laboratory (MDOlab) in the Department of Aerospace Engineering. He is currently on sabbatical leave as an invited professor and Marie Curie Fellow at the Institut Supérieur de l'Aéronautique et de l'Espace (ISAEâĂŞSupAéro). His research involves the development and application of MDO methodologies to the design of aircraft configurations, with a focus on high-fidelity simulations that take advantage of high-performance parallel computing. Before joining the University of Michigan faculty in September 2009, he was an Associate Professor at the University of Toronto, where from 2002 he held a Tier II Canada Research Chair in Multidisciplinary Optimization. Prof. Martins received his undergraduate degree in Aeronautical Engineering from Imperial College, London, with a British Aerospace Award. He obtained both his M.Sc. and Ph.D. degrees from Stanford University, where he was awarded the Ballhaus prize for best thesis in the Department of Aeronautics and Astronautics. He was a keynote speaker at the International Forum on Aeroelasticity and Structural Dynamics (2007), Aircraft Structural Design Conference (2010), and SIAM Conference on Optimization (2014). He has received the Best Paper Award in the AIAA Multidisciplinary Analysis and Optimization Conference four times (2002, 2006, 2012, and 2014). He is a member of the AIAA MDO Technical Committee and was the technical co-chair for the 2008 AIAA Multidisciplinary Analysis and Optimization Conference. He is also an Associate Editor for Optimization and Engineering and Structural and Multidisciplinary Optimisation.

Abstract of the presentation

The wing is a crucial aircraft component, and its shape has a large impact on performance. Wing design optimization has been an active area of research for several decades, but achieving practical designs has been a challenge. One of the main challenges is the wing flexibility, which requires the consideration of both aerodynamics and structures. To address this, we proposed the simultaneous optimization of the outer mold line of a wing and its structural sizing. The solution of such design optimization problems is made possible by a framework for high-fidelity aerostructural optimization that uses state-of-the-art numerical methods. This framework combines a three-dimensional CFD solver, a finite-element structural model of the wingbox, a geometry modeler, and a gradient-based optimizer. With this framework we are able to compute the flying shape at the various flight conditions, and to optimize aircraft configurations with respect to hundreds of aerodynamic shape and internal structural sizes. The theoretical developments include coupled-adjoint sensitivity analysis, an automatic differentiation adjoint approach, and new formulations for various practical constraints. The algorithms resulting from these developments are all implemented to take advantage of high-performance parallel computing. Applications to the optimization of aircraft configurations demonstrate the effectiveness of these approaches in designing aircraft wings for minimum fuel burn. The results show optimal tradeoffs with respect to wing span and sweep, which was previously not possible with high-fidelity models.

Design for Additive Manufacturing through Topology Optimization

Uwe Schramm* and Ming Zhou†
Altair Engineering Inc., 38 Executive Park, Irvine, CA 92614.

ADDITIVE manufacturing has a long history. In recent years certain additive manufacturing technologies have gained a new attention. This is mostly driven by the need for light weight design, mass customization, as well as rapid manufacturing. The promise of additive manufacturing is form freedom, complexity made easy, individualized products, and accelerated production.

On the other hand there has been a shift in how products are designed. More and more design processes are driven by the use of computer simulation rather than physical experimentation. Technologies like Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) have been transformed from being purely used for product validation and failure mode analysis to being design tools. In particular Topology Optimization has emerged as a powerful simulation-based design tool. Topology Optimization uses load information to derive the optimal material layout for light weight design. These designs have a very aesthetic look and remind us often about biological designs. Topology Optimization can help engineers to think out of the box during early stage of concept generation. It can generate surprisingly efficient designs in engineering fields where products are already considered highly sophisticated. For example, Airbus achieved over 40% weight reduction by applying topology optimization on a group of A380 leading edge ribs. Since topology optimization often creates free-forming 'bionic' structures, interpretation of design concept that fits traditional manufacturing methods has been a challenge.

There seems to be a natural symbiosis between simulation-driven design and additive manufacturing. Often the true optimal design cannot be manufactured with other means like casting, forging, milling, or extrusion. Then manufacturing constraints have to be introduced in the concept phase of the design where topology optimization is used. With additive manufacturing, there are less such constraints and the manufactured part can be closer to a theoretical design optimum. Additive manufacturing brings almost unlimited freedom for design shape and form, hence offers the perfect combination with topology optimization for creation of most efficient structures.

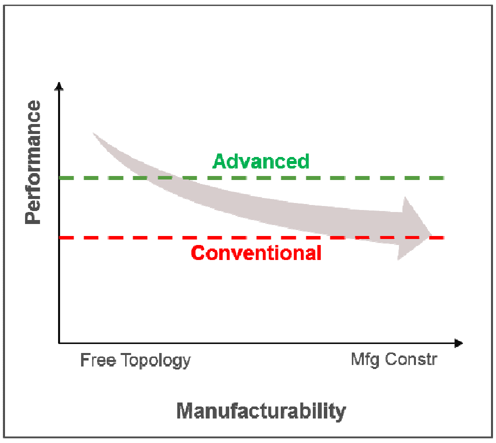


Figure 1: Performance of design conceived for advanced vs. conventional manufacturing.

An additive manufacturing technique that has gotten the most recognition lately is 3D-Printing. Both are used as synonyms these days. There are other techniques that could also be considered additive such as composite layups. Latter also has the ability to produce structures that follow the load path and as such give optimum performance with little or no constraints.

Communicating Author: schramm@altair.com
* CTO.
† Senior Vice President.

One unique capability of 3D-Printing is that it enables creation of structures built with tiny members, known as lattice structure. Lattice structure optimization is a novel solution to create blended solid and lattice structures from concept to detailed final design. It has been implemented in the industry leading optimization software Altair OptiStruct [1]. This technology has been developed in particular to assist design innovation for additive manufacturing. The solution is achieved through two optimization phases. In Phase I classic Topology Optimization is performed, albeit reduced penalty options are provided to allow more porous material with intermediate density to exist. Phase II transforms porous zones from Phase I into explicit lattice structure. Then lattice member dimensions are optimized in the second phase, typically with detailed constraints on stress, displacements etc. The final result is a structure blended with solid parts and lattice zones of varying material volume. Currently two types of lattice cell layout are provided: tetrahedron and pyramid/diamond shaped cells. This results in smoother shape of lattice structure, while also offers flexibility in controlling lattice cell size independent of the initial mesh.

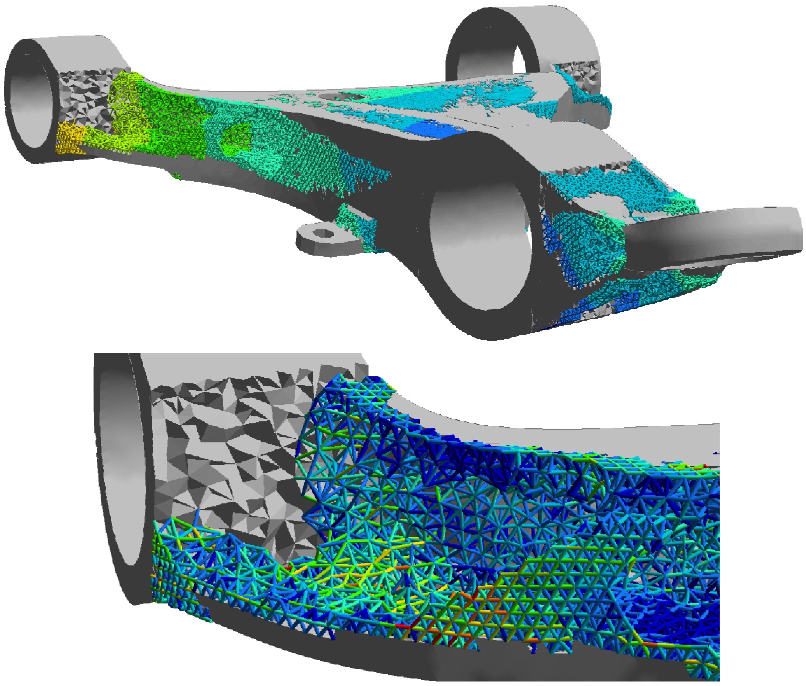


Figure 2: Suspension control arm: Final structure blended with solids and lattice.

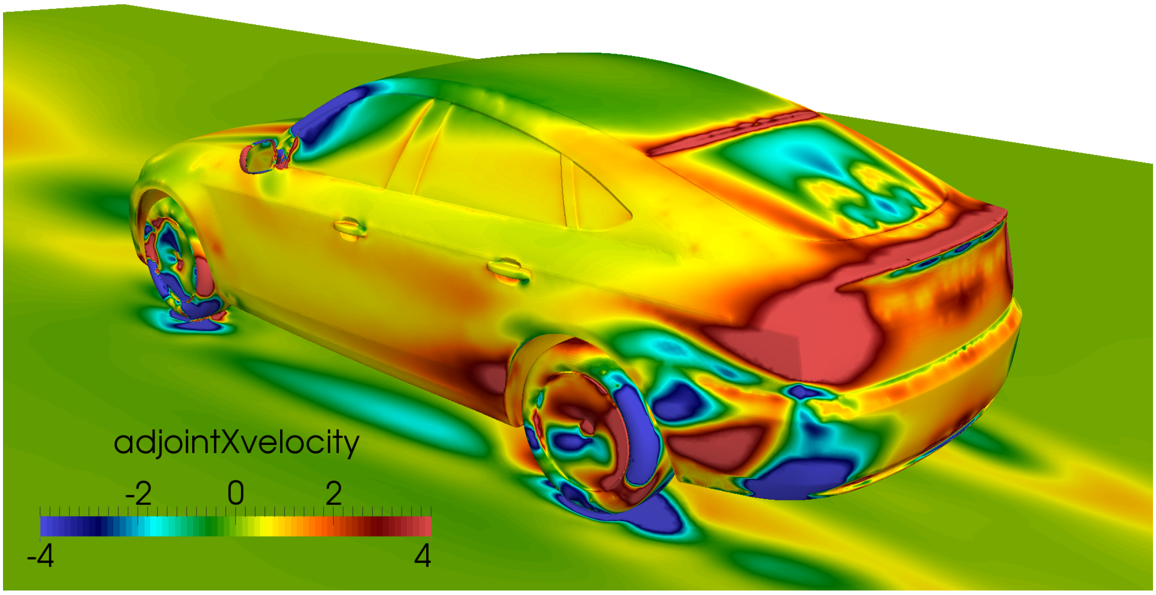
Acknowledgements

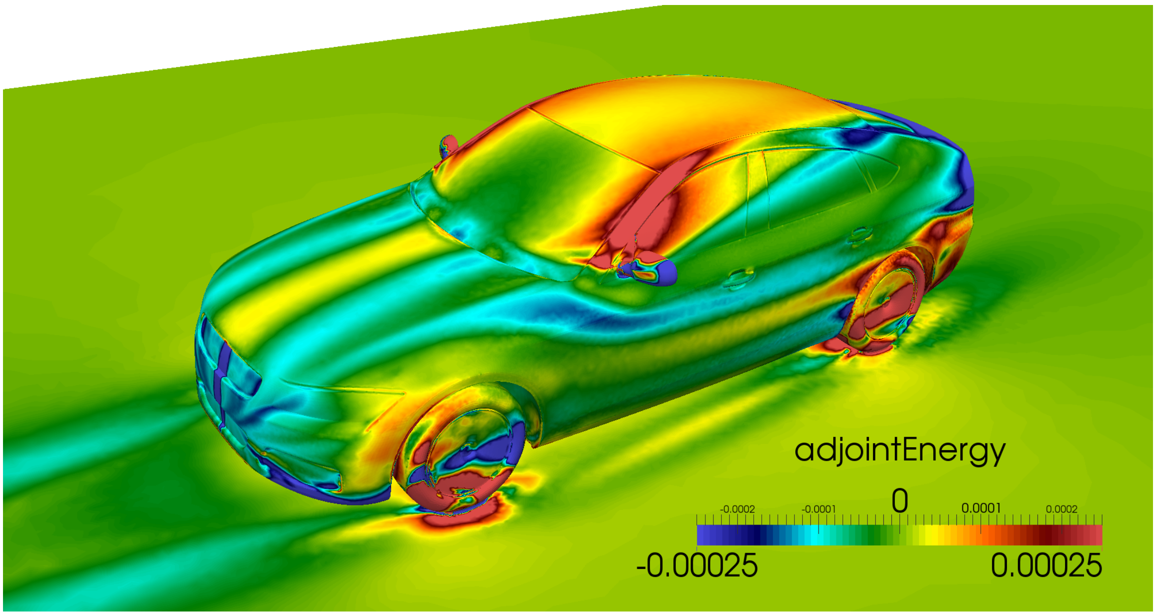
The authors wish to extend their appreciation to all members of the Altair software and business development organization as well as Altair Product Design for their continued innovative approach to developing and implementing structural optimization technology. Many thanks also to our customers for implementing these techniques, sharing their experience, and helping us learn about their work. This is a true and worldwide team-effort.

References

[1]Altair OptiStruct 14.0 Users' Manual, Altair Engineering, Troy, MI, 2015.

3.2 Contributed Presentations





Adjoint solution for drag around the DrivAer vehicle computed with STAMPS of QMUL. Top: adjoint velocity, bottom: adjoint energy. Meshes have been kindly provided by Beta CAE, www.beta-cae.com.

Computational Design for Micro-Fluidic Devices using a Tightly Coupled Lattice Boltzmann and Level Set-Based Optimization Algorithm

David J. Munk* and Gareth A. Vio†
The University of Sydney, Sydney, 2006, Australia

and

Timoleon Kipouros‡ and Geoff T. Parks§
University of Cambridge, Cambridge, CB2 1PZ, the United Kingdom

Recently the study of micro-fluidic devices has gained much interest in various fields from biological to engineering. The reason for this increased interest can be attributed to the technological progress in machining, allowing complex geometries to be manufactured, and the need for miniaturized devices in combustion and chemical analysis. These devices are concerned with low Reynolds number flows, resulting in a laminar flow regime, having dimensions that range from millimeters to micrometers. Although micro-fluidic devices can be implemented for mixing of multiple fluid species; this work is particularly interested in the mixing of two non-reacting iso-thermal and incompressible fluids. Therefore, whilst diffusivity dictates the mixing in low Reynolds number flows, in this case the mixing is governed only by turbulence. Thus, the mixing can be enhanced by using active or passive devices, such as moving parts or multi-holed baffle plates. In this study passive devices are considered, due to their ease of integration and stable operation, compared with active devices, which require an energy input in order to mix the flows. Complex geometries are required due to the presence of components in the direction of the flow which stretch and fold the fluid over the cross-section of the channel enhancing the turbulence of the system. Hence, areas of high stress are present due to the geometrical non-linearity of the micro-fluidic devices and high pressure ratios. This article develops a novel framework for a topology optimization algorithm that is coupled directly to the Lattice Boltzmann method, used for simulating the flow in the micro-fluidic device, for the objective of minimum compliance. This study focuses on the effect of the fluid-structural interactions by comparing the optimization results obtained by a fully coupled, where the loads are updated directly by the change in topology, and uncoupled, no update in the load, solutions on the design of micro-fluidic devices. The final compliance for both cases are compared and a trade-off is made between minimum compliance and computation time.

I. Computational Methodology

The numerical framework, which couples the topology optimization algorithm with the Lattice Boltzmann flow solver, is shown in Figure 1. The problem is defined, setting the initial topology and boundary conditions for the test case. The geometry is passed onto the flow solver, which outputs the pressures, and ultimately the forces, being applied to the topology. The current topology and loads are passed onto the Finite Element Method (FEM) module, which defines the structural boundary conditions and outputs the compliance of the structure. This is passed onto the topology optimization algorithm which calculates the sensitivities and updates the topology of the structure for the next iteration.

The numerical framework (Fig. 1) consists of two loops. The first is between the topology optimization algorithm and the FEM module, which is performed at every iteration. The second loop passes the updated topology back into the LBM flow solver. Due to the computational penalty of the LBM solver, the second loop is performed only after every n_{LBM} iterations. This is a predefined parameter, which ensures the topology has

Communicating Author: david.munk@sydney.edu.au
* Research Student, AMME, the University of Sydney, NSW, 2006, Australia.
† Senior Lecturer, AMME, the University of Sydney, NSW, 2006, Australia
‡ Research Fellow, EDC, University of Cambridge, Cambridge, CB2 1Pz, the United Kingdom
§ Reader, EDC, University of Cambridge, Cambridge, CB2 1Pz, the United Kingdom

changed significantly, such that the loads are required to be updated. A trade-off between reduction in compliance and computation time is given in the results.

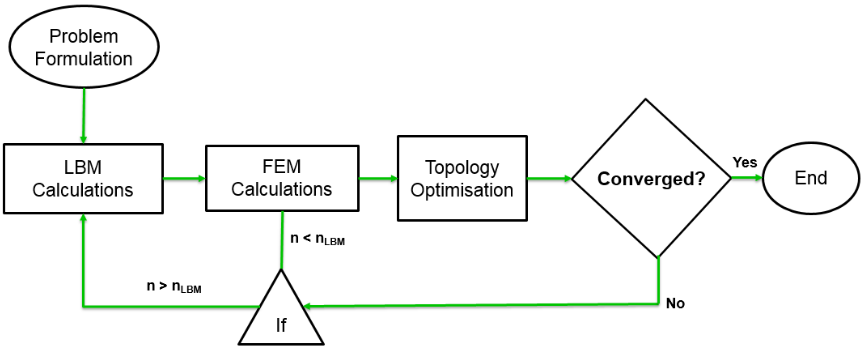


Figure 1. Numerical Framework for Coupled LBM-Topology Optimization.

II. Preliminary Results

In this section the preliminary results of the numerical framework are presented, along with a discussion and summary of the findings of this study. The preliminary results for the uncoupled and coupled algorithm are given.

First, the problem with no feedback to the fluid solver is analyzed. Therefore, the LBM is only performed once on the initial structure to obtain the pressure loads, which remain unchanged throughout the optimization. This represents the simplest and hence most computationally efficient case. Next, the coupled solution is presented, using a $n_{LBM} = 10$, with all other parameters identical to the uncoupled solution.

The initial and final topology for the uncoupled and coupled problem is shown in Figure 2.

Figure 2. Initial Topology (left) Final Topology Uncoupled (middle) Final Topology Coupled (right).

The final topology shows a much more complex structure (Figure 2), creating load paths to increase the stiffness of the baffle. All six holes from the original structure have been maintained; however, the topology of the holes differ significantly. Further, four smaller holes have been added to the baffle. Likewise, The final topology for the coupled case (Figure 2 (right)) differs significantly compared to the uncoupled solution (Figure 2 (middle)). Namely, the coupled solution has added two larger holes along the mid horizontal plane, which are not present in the uncoupled case. Further, the holes have become less triangular, reducing the sharp corners in the topology. The strain energy distribution on the initial, uncoupled and coupled topology is given in Figure 3.

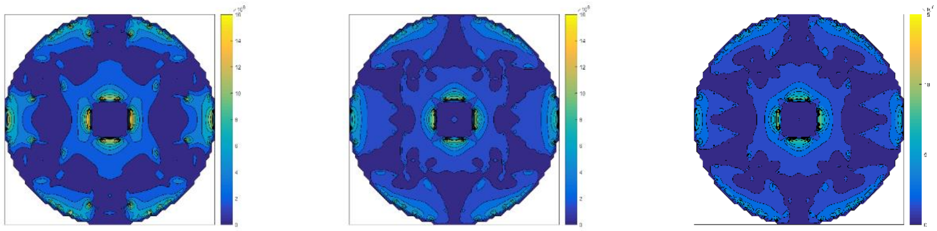


Figure 3. Initial Compliance (left) Final Compliance Uncoupled (middle) Final Compliance Coupled (right).

The initial strain energy distribution shows several concentrated regions around the holes, where the overall stress is significantly higher compared to the rest of the structure. Comparatively, the optimized topology significantly reduces these concentrated zones (Figure 3), especially if the center hole is ignored, since this region is non-designable. The coupled solution further reduces the strain energy concentrations, compared with the uncoupled solution.

Topological Optimal Design and Experimental Validation of a Milled Liquid Cold Plate

Dr. ir. J. Coddé * and dr. ir. L. Vervecken†
Diabatix bvba, Kapeldreef 60, 3000 Leuven, Belgium

This paper details the design of a liquid cold plate by topology optimization. The topology optimization procedure minimizes for a uniform surface temperature. The design is constrained by its assembly with screws, the need for structural integrity and the manufacturing process, CNC milling. The resulting cold plate is used as a replacement for a cold plate with a serpentine cooling channel. Both cold plates are experimentally tested in a setup which represents a general industrial use case. These experiments demonstrate that the average thermal resistance between the coolant and the heat sources can effectively be lowered by more than a factor of 2 by the application of topology optimization.

I. ....Introduction

THE cooling of machine components is a recurring problem in industry. One reason for this is that, very often, the operating temperature of these components is inversely proportional to its lifetime. Therefore, active cooling is applied to the components in an attempt to keep the maximal temperature below a certain limit temperature. Examples of such components range from lasers to engines and power electronics. Also products may require cooling during production. This is the case for instance for casting. By applying active cooling during casting, the cycle time is reduced with an increased productivity as a result. However, inefficient or non-uniform cooling can result in a reduced product quality. The goal of this paper is to explore the benefit of new cooling solutions enabled by topology optimization in an industrial setting.

This paper focuses on a generic liquid cold plate. A liquid cold plate is essentially a heat sink through which a liquid coolant flows. A number of discrete heating elements, mounted to the cold plate, serve as heat sources. The common design for such a generic cold plate is a CNC-milled metal plate with one serpentine channel which runs over all heating elements. In this work, a new design for the CNC-milled cold plate is generated using topology optimization. The goal function is a uniform surface temperature, thus considering both flow and heat transfer. Thereby, all necessary constraints are taken into account: assembly constraints (screws), manufacturing constraints, structural integrity and a limited pressure drop.

The new design is manufactured and tested against the original design on a dedicated test setup. The test setup determines has a range of operating conditions. The optimal design is created based on the worst case conditions (lowest mass flow etc.). Therefore, the robustness of the design with respect to variations in the operational conditions is also subject of the experimental validation.

Subsequently, the cold plate with a serpentine cooling channel and relevant design specifications, the design of a new cold plate using topology optimization and the experimental validation are discussed briefly.

II. ... Cold Plate with Serpentine Cooling Channel and Design Specifications

Nearly all designs of liquid cold plate are constructed from multiple equal-sized parallel channels or a serpentine cooling channel. In this case, the reference design is with the serpentine channel, pictured in Figure 1a. It is clear from this figure that the serpentine channel structure is projected onto the surface temperature for the given operating conditions. Moreover, in fig. 1b a cross-section of the coldplate is shown, which demonstrates the heating of the coolant along the channel. Clearly, depending on the operating conditions, this can greatly aggravate the effective cooling of the heaters near the end of the cooling channel.

In the test setup, a number of discrete heating elements is attached to the cold plate. These heating elements are cooled by glycol running through the cold plate. The temperatures of the heaters are measured and serve as the benchmark for the topologically optimized design. Since the cold plate is milled, a thermally isolating plastic cover is mounted to the back of the to close off the channels. These design specifications translate into boundary conditions and constraints for the optimal design detailed in the next section.

Communicating Author: joris.codde@diabatix.com.
* Chief Technology Officer.
† Chief Executive Officer.

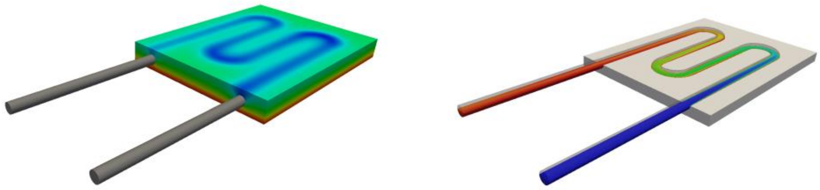


Figure 1. Thermal view of the liquid cold plate, revealing the serpentine cooling channel.

III..... Topology optimization of a Liquid Cold Plate

A new, optimal design of the liquid cold plate is generated using topology optimization with the density approach. This requires the solution of the Navier-Stokes equations, including the energy equation, in both forward and adjoint formulation. Because the flow in the heat sink is laminar, no turbulence modeling is required. The pressure drop from inlet to outlet is set equal to the pressure drop of the original cold plate. The goal function is a uniform surface temperature of the contact area between the heaters and the cold plate:

$$\min J(\varphi,\varepsilon) = \int_{\Omega_h} (T(\vec{x}) - T_{ref})^2 d\vec{x}$$

where J is the goal function, φ the state, ε the design variable (1 = fluid phase, 0 = solid phase), Ω_h interface between cold plate and heaters, T the temperature, T_{ref} the reference temperature and $\vec{x}$ the coordinate vector. The resulting optimization problem is similar to in [1] and will not be repeated here.

In order to ensure the manufacturability and the usability of the resulting design, the following additional constraints are imposed:

- Assembly screws: the heaters are mounted on the cold plate with screws. All cells within a radius of these screws are constrained to stay solid (ε = 0).
- Structural integrity: the rim of the plate as well as the bottom of the plate (where the heaters are attached) is constrained to the solid phase (ε = 0). This is a rough way of asserting structural integrity. However, these conditions were extracted from the original design with the serpentine cooling channel and thus mainly limit the freedom of the topology optimization. Refining this constraint is a future task.
- Manufacturability: channels that are cut out in the plate all have the same depth (equal to the height of the plate minus the height of the fixed bottom imposed in the previous constraint). This ensures that channels can be cut out and that no hollow structures are formed.

The final design consists of a network of cooling channels which ensures maximal local cooling and minimal overall pressure drop.

IV. ....Experimental Validation

The experimental validation is ongoing, and will test the liquid cold plate designed in this work versus the original cold plate with serpentine cooling channel. The results will be presented at the conference. Based on the simulations, it is expected that the average thermal resistance between heaters and fluid will lower by a factor 2.

V. ....Conclusion

In this work, a liquid cold plate was designed by topology optimization and will soon be experimentally validated. The liquid cold plate was optimized towards a uniform temperature of the attached discrete heating elements. The design is constrained by its assembly with screws, the need for structural integrity and the manufacturing process, CNC milling. The final design consists of a network of cooling channels which ensures maximal local cooling and minimal overall pressure drop.

The experimental validation is ongoing, and will test the liquid cold plate designed in this work versus the original cold plate with serpentine cooling channel. Based on the simulations, it is expected that the average thermal resistance between heaters and fluid will lower by a factor 2.

References

Qian, X., & Dede, E. M., Topology Optimization of a Coupled Thermal-Fluid System under a Tangential Thermal Gradient Constraint. *Structural and Multidisciplinary Optimization*, 2016.

DEVELOPMENT OF AN ACCURATE PROCESS WORKFLOW FOR SIMULATION OF A GAS LIQUID CYCLONE SEPARATOR IN AN OPEN-SOURCE ENVIRONMENT

ANIMESH RANJAN[1], SWAMI VENKAT[2],ROMIT MAULIK[3] , FLAVIU SIMON[4]

[1]Technische Hochschule Ingolstadt, animesh.ranjan@gmx.net
[2]Numerics GmbH, swami.venkat@numerics.de
[3]Oklahoma State University, romit.maulik@gmail.com
[4]Bauhaus University, flaviu.bogdan.simon@uni-weimar.de

Keywords: *Cyclone Separator, 3D Fluid Flow Simulation, Fluid-flow Optimization*

Cyclone separation is a technique to separate particulate matter or a given gaseous phase from a fluid jet using rotational effects and gravitational forces. They are employed in sawmills, oil refineries, cement industries and is also the fundamental principle of several everyday appliances such as vacuum cleaners. The widespread utilization of Cyclone separators has prompted a deeper understanding of an otherwise complex phenomena. With the advent of advanced CFD approaches and enhancement of computational resources, fluid flow in a cyclone separator can be understood beyond basic analytical methods such as Stokes' law. The paper discusses the intricacies of generation of a process work-flow for 3D fluid flow simulation in a Gas Liquid Cylindrical Cyclone Separator (GLCC) using open-source environment such as Salome, OPENFOAM and Paraview. The main idea is to generate a robust work-flow which addresses the uncertainties in the boundary conditions, turbulence models and other flow parameters, thereby seeking to achieve improved correlation between experiment and CFD simulation. The process includes several stages of complexity identification, model evaluation, simplified model development and Best FIT design identification. The overall results show highly accurate agreement between experiment and simulation and could pave way for greater understanding of the physical phenomena in GLCC separator. Moreover, the implemented methodology is designed to serve as a benchmark for design development and flow optimization studies.

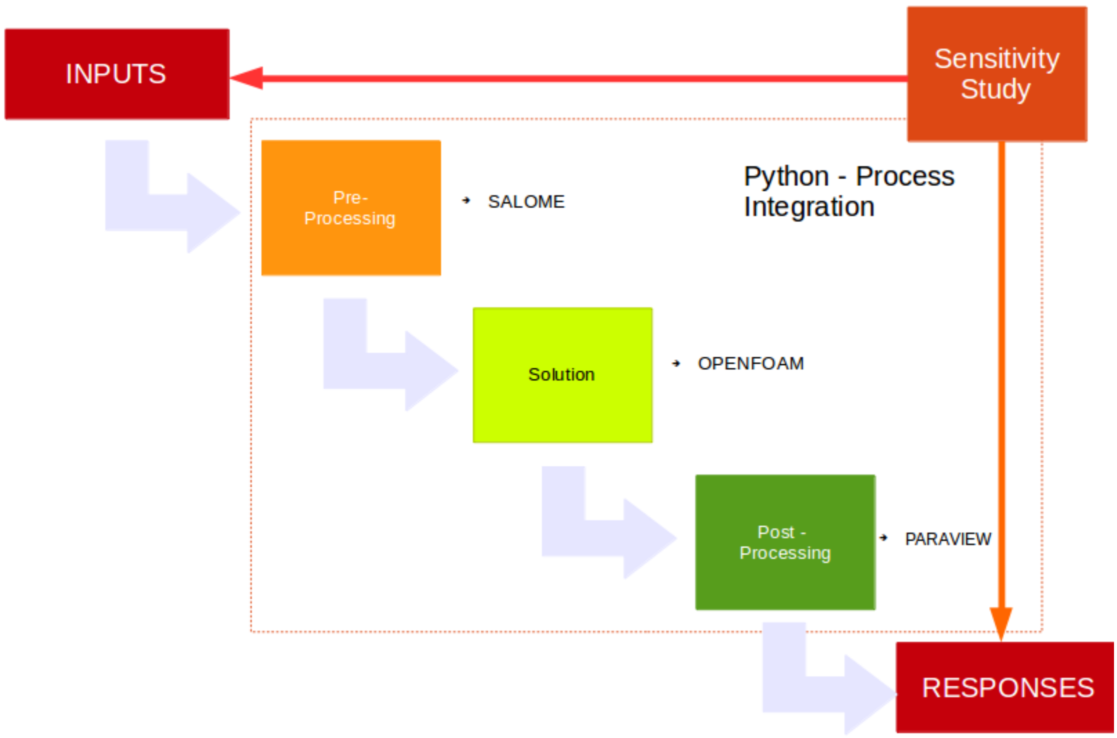


Figure 1: Process Flow Chain for 3D GLCC Simulation

Optimal design of a heat dissipating wheel for a 2CV race car

Nick Van Kelecom[*] and Tom Wellens[†]
Faculty of engineering technology, Campus Group T, Leuven, Vesaliusstraat 13, 3000 Leuven, Belgium

Yann Caniou[‡] and Maximilien Landrain[§]
Noesis Solutions NV, Gaston Geenslaan 11 B4, 3001 Leuven, Belgium

and

Maarten Vaniershot[**]
Faculty of engineering technology, Campus Group T, Leuven, Vesaliusstraat 13, 3000 Leuven, Belgium

The field of race cars is the ideal place for innovation that will be later proposed to the public vehicles. This work deals with the issue of the need of a race car engine to be able to dissipate the heat create by the fuel combustion in order to increase its level of performance. The solution proposed here consists in developing competition turbine shaped rims that would be able to extract the air from under the bonnet of the car. A numerical model is created in order to simulate both the air flow and the mechanical behavior of the rim. The initial design is finally optimized in order to maximize the air extraction and to minimize the weight of the rim.

Nomenclature

CFD	=	Computational Fluid Dynamic
FEA	=	amplitude of oscillation

I. Abstract

The CQS Group Racing Team recently came up with a problem: their engine needs to work in optimal conditions to deliver its best performance during the competition. One of these conditions is to reduce as much as possible the amount of hot air under the bonnet so that the engine is able to dissipate its heat with fresh air.

This research work investigates the possibility of designing a rim that is able to extract the hot air from underneath the hood of a 2CV race car. The main advantage of this kind of rims is that the engine is able to cool down faster than with a natural extraction of the hot air. In order to make it possible, the spokes of the rim are designed like the blades of a turbine so that the air can be extracted. The mass flow was determined by Computational Fluid Dynamics (CFD) modelling on the extracted air.

For safety reasons, another feature that has required a lot of attention alongside the CFD is a Finite Element Analysis (FEA) structural analysis of the rim. The objective of this second analysis was to assess the resistance of the rim design to mechanical loads. The structural loads that are applied on the rim are the centrifugal force, the pressure of the tire and the weight of the car. Afterwards, a parametric numerical model was created and used in an optimization process whose objective was to maximize the air extraction and minimize the mass of the rim with respect to structural constraints.

The optimization strategy that has been implemented is a classical approach. An adjustable full factorial design of experiments is run in order to capture the behavior of the model with metamodel, also referred to as response surface, namely a polynomial representation here. After the accuracy metamodel has been assessed using cross validation, a two-step optimization strategy has been applied. First a global optimization method,

Communicating Author: yann.caniou@noesissolutions.com.
[*] Master of industrial science, KU Leuven, Vesaliusstraat 13, 3000 Leuven, Belgium.
[†] Master of industrial science, KU Leuven, Vesaliusstraat 13, 3000 Leuven, Belgium.
[‡] Application Engineer, Noesis Solutions, Gaston Geenslaan 11 B4, 3001 Leuven, Belgium.
[§] Application Engineer, Noesis Solutions, Gaston Geenslaan 11 B4, 3001 Leuven, Belgium.
[**] Professor, KU Leuven, Vesaliusstraat 13, 3000 Leuven, Belgium.

namely an evolutionary algorithm (EA), has been run to identify a region of interest. An EA is an optimization algorithm that mimics biological phenomena: starting from an initial population of individuals, it iteratively applies mechanisms such as selection, mutation, recombination and reproduction to propose new candidates that are evaluated using a fitness function. Then a local optimization method, namely a gradient-based algorithm has followed to reach an optimal design point. Finally, this possibly global optimum has been validated by a CFD and a structural analysis to confirm the results. This entire strategy has been carried out inside the Optimus® platform. This tool allows the practitioner to capture the process, here both the structural and CFD analyses, in a workflow and to automate all the above-mentioned steps of the strategy.

The results show that it is possible to develop such a rim that is light but string enough to withstand the solicitations of a race car and that is able to extract the hot air from the engine bay of the 2CV. The results obtained through the structural analysis are comparable with those obtained with the previously run manual calculations. The boundary conditions that were placed on the model for the flow analysis were “moving frame of reference”, “static pressure type” and “boundary flow surface”. The flow analysis has shown good velocity and pressure results. Future improvements could be further enhanced by decreasing the mesh size.

Aeroelastic tailoring for gust load alleviation

Paul Lancelot* and Roeland De Breuker†

Faculty of Aerospace Engineering, Delft University of Technology, The Netherlands

Key words: aeroelasticity, structural optimisation, dynamic load, load alleviation, composite material

Summary: In this paper, the structural optimisation process of a wing that is designed for gust load alleviation is presented. The optimisation process is built around the equivalent static loads method and Nastran.

Introduction:

Load alleviation has been a field of research which has received more and more attention over the past decade due to the development of light-weight highly flexible wings used in modern airliners as well as for high altitude long endurance (HALE) uninhabited aerial vehicles (UAVs). It has been identified as an efficient way to reduce structural fatigue and to improve aircraft handling as well as passenger comfort. Instead of active load control using ailerons and spoilers which has been around since forty years, composite materials can also be used to obtain a flexible wing that can deform in such way that it will passively relieve itself from the loads. But flexibility also means that unwanted interactions can occur between the control surfaces, leading to control efficiency problems. In this paper, the structural optimisation process of a wing that is designed for passive gust load alleviation and also complies with constraints related to structural strength and stiffness, aeroelastic instability (flutter, divergence) and minimum control effectiveness over the entire flight envelope, is presented. To perform this optimisation, a gradient-based approach is preferred as the number of design variables is relatively large (> 100). However the required sensitivity computation over a transient response is not an easy task [1]. The equivalent static loads (ESL) method recently formalised by Park [2] is used to bypass this issue, and has already been applied to similar problems [3], [4]. It is combined with existing aeroelastic and structural optimisation framework, and can be extended to non-linear cases [5]. Although this technique only provides a weak coupling between the design variables and the loads, its main advantage resides in its ease of implementation.

Preliminary results:

Preliminary results show that the ESL method can provide accurate static loads to the optimisation routine by reproducing the strain field from the dynamic simulation into a static case. This is sufficient to achieve convergence within the required constraints. For this first study, a transport aircraft wing that can freely move in plunge is hit by a “1-cosine gust”. The objective function is the structural mass and only the strength constraints are applied. The loads are extracted from the transient aeroelastic NASTRAN solution [6] while the sizing is performed within the NASTRAN optimisation module [7]. A custom-built MATLAB script is used for the data transfer between the two solvers. Thicknesses of several panels of the wing box are used as design variables, and the convergence is reached after 25 iterations, providing a significant reduction root bending moment and weight saving (see Figure 1).

* PhD Candidate; P.M.G.J.Lancelot@tudelft.nl

† Assistant Professor; R.DeBreuker@tudelft.nl

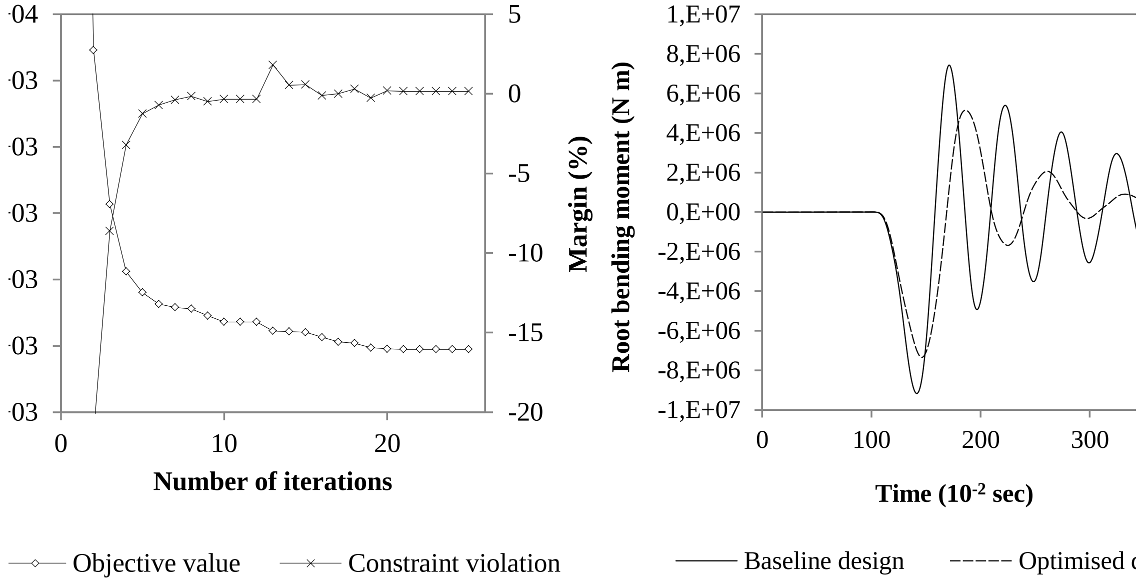


Figure 1: Optimisation results of a wing hit by a 1-cos gust.

In order to assess the limitations of the ESL methodology and the number of iterations required to reach convergence, test cases with different gust scenarios, wing geometries and boundary conditions will be performed (clamped or free pitch and/or plunge). Design variables will be extended to composite laminate properties and additional constraints will be added regarding aeroelastic instability and control effectiveness. The results will be presented in the completed paper. Finally, a trade-off study will be introduced between wing flexibility and effectiveness of different load alleviation strategies.

References:

[1] D. Bettebghor and C. Blondeau, “Prise en compte de la flexibilité des cas de charges dimensionnants en optimisation de structure,” presented at the CSMA 2013 11e Colloque National en Calcul des Structures, 2013.

[2] G.-J. Park, “Technical overview of the equivalent static loads method for non-linear static response structural optimization,” *Struct. Multidiscip. Optim.*, vol. 43, no. 3, pp. 319–337, Jul. 2010.

[3] F. Daoud, O. Petersson, S. Deinert and P. Bronny, “Multidisciplinary Airframe Design Process: Incorporation of steady and unsteady aeroelastic loads,” in *12th AIAA Aviation Technology, Integration, and Operations (ATIO) Conference and 14th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, American Institute of Aeronautics and Astronautics.

[4] A. Wildschek, T. Haniš, and F. Stroscher, “L_∞ -Optimal feedforward gust load alleviation design for a large blended wing body airliner,” 2013, pp. 707–728.

[5] Y. I. Kim, G. J. Park, R. M. Kolonay, M. Blair, and R. A. Canfield, “Nonlinear Dynamic Response Structural Optimization of a Joined-Wing Using Equivalent Static Loads,” *J. Aircr.*, vol. 46, no. 3, pp. 821–831, 2009.

[6] MSC.Software Corporation, *MSC Nastran 2014 Aeroelastic Analysis User’s Guide*. 2014.

[7] MSC.Software Corporation, *MSC Nastran 2012 Design Sensitivity and Optimization User’s Guide*. 2012.

Blending Constraints in Composite Wing Aeroelastic Optimization

Marco Tito Bordogmna* and Dimitri Bettebghor†
ONERA - The French Aerospace Lab, 92260, Châtillon, France

Roeland De Breuker‡
Delft University of Technology, 2629 HS Delft, The Netherlands

Improving large composite structure can be achieved by locally optimizing thicknesses and ply orientations. However, this strategy can lead to structural discontinuity or unfeasible structure from the manufacturing point of view. Therefore, enforcing blending during structural optimization is fundamental in order to achieve ready-to-manufacture solutions. The present paper is expected to prove the effectiveness of blending constraints in the aeroelastic optimization of a variable stiffness wing. The paper should prove that more realistic optimal continuous design can be achieved due to the application of the blending constraints. Moreover, such constraints will be implemented in the commercial software Nastran SOL 200 demonstrating their ease of implementation.

I. Introduction

Composite materials are being used more and more often in the aerospace industry thanks to their high stiffness and strength to weight ratios when compared to the more conventional aluminum counterpart. The anisotropic properties of composite, together with the possibility of designing the laminate stacking sequence, allows aircraft designers to tailor the stiffness of composite structures according to the loads they are predicted to experience. Improving stiffness design of a large composite structure can be obtained by dividing the structure in sections and locally optimizing thickness and stacking sequence in each of the section. However, this might lead to stress concentration between adjacent sections in case of thickness and/or stacking sequence discontinuities. In order to avoid this stress concentration, blending has to be taken into account in order to ensure ply continuity for the whole structure.

Ply continuity can be imposed in different ways. Inner and outer blending have been introduced by Adams et al.[1] , in these definitions only the innermost and the outermost plies can be dropped as shown in Figure 1a. Two alternative definitions, the generalized and relaxed generalized blending, have been formulated by Van Campen et al.[2] and are presented in Figure 1b. Generalized blending requires all plies of the thinnest section to be continuous in the whole structure; relaxed generalized blending demands that no discontinuous plies should be in direct physical contact with each other. Throughout this paper, blending is always associated to the generalised blending definition of Van Campen et al.[2] for sake of clarity.

II. Methodology

Several authors[3456] have used bi-step strategies where a gradient based (continuous) optimization of homogenized stiffness parameters (e.g. lamination parameters) is followed by genetic algorithm (discrete optimization) to retrieve blended stacking sequence. By doing so, mechanical constraints are verified during the continuous step while most of the manufacturing constraints (e.g. blending) are enforced only in the discrete step. This can result in significant discrepancies between the two optimization steps[7] and therefore there is no guarantee to find an equivalent of the optimal continuous design after the discrete

*PhD Candidate, DADS/MSAE, 29 Avenue de la Division Leclerc, 92320 Châtillon.
†Researcher, DADS/MSAE, 29 Avenue de la Division Leclerc, 92320 Châtillon.
‡Assistant professor, Faculty of Aerospace Engineering, ASCM, 2629 HS Delft, The Netherlands.

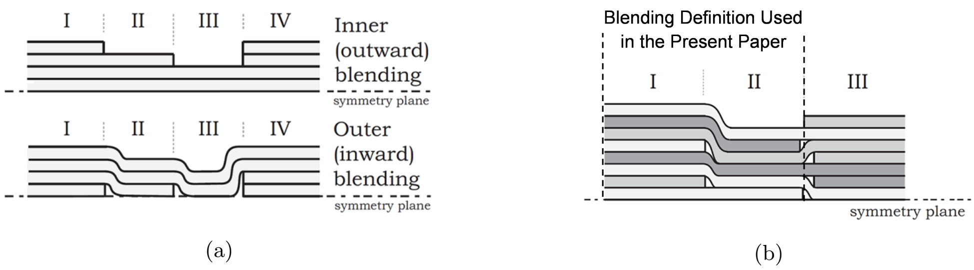


Figure 1: (a) Outward and inward blending, and (b) generalised (I and II) and relaxed generalised (II and III) blending. Original figures from.[2]

step.

In view of the above, Macquart et al.[8] have derived a set of continuous blending constraints in order to achieve more realistic continuous design and reduce discrepancies between the two optimization steps. The blending constraints have been derived for lamination parameters. The key concept for the derivation of the continuous blending constraints is to evaluate the change in lamination parameters (ΔV) due to ply drops. Blending constraints have been derived for both in-plane and out-of-plane lamination parameters.

III. Results

This paper wants to investigate the effectiveness of continuous blending constraints in the aeroelastic optimization of a variable stiffness composite wing (Figure 2). The paper considers the effect of blending constraints relaxation and grouping. Finally, to prove that the optimization has converged to a reasonable improved design, a multi start strategy is applied to ensure the robustness of the optimization. Following the continuous optimization a stacking sequence table based GA is used to retrieve a blended stacking sequence. Since the blending constraints reduce the design space to more realistic solutions and to avoid the use of expensive fitness function, the GA objective is to match the lamination parameters of the optimal continuous design. The optimization is carried out with *MSC Nastran SOL 200* proving that the constraints can be easily implemented in commercial optimizer.

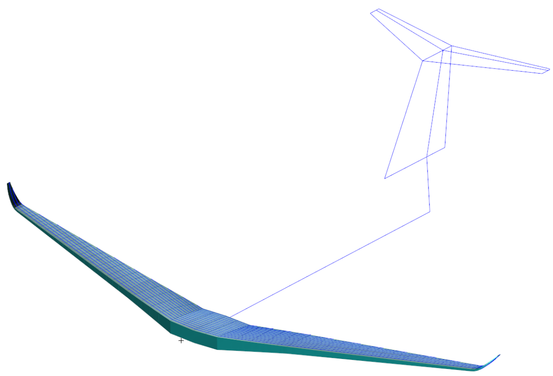


Figure 2: Aircraft wing model used in the optimization.

References

[1]Adams DB, Watson LT, Gurdal Z, Anderson-Cook CM. *Genetic algorithm optimization and blending of composite laminates by locally reducing laminate thickness.* Advances in Engineering Software. 2004;35:35-43.

[2]Van Campen J, Seresta O, Abdalla MM, Gurdal Z. *General blending definitions for stacking sequence design of composite laminate structure.* Proceedings of 49th AIAA/ASME/ASCE/AHS/ASC structures, structural dynamics, and materials conference, Schaumburg, IL, USA2008. p. 7-10.

[3]Peeters DM, Hesse S, Abdalla MM. *Stacking sequence optimisation of variable stiffness laminates with manufacturing*

Multi-fidelity aeroelastic analysis and sensitivity analysis for gradient-based structural optimization

Kristofer Jovanov, * Roeland De Breuker, † and Mostafa M. Abdalla ‡

Delft University of Technology, 2629 HS Delft, The Netherlands

Christophe Blondeau, §

ONERA BP72 - 29 avenue de la Division Leclerc, FR-92322 Chatillon Cedex, France

In this paper a linear aerodynamics-based preconditioner is presented with the capability of reducing the solution time of high-fidelity aeroelastic analysis and sensitivity analysis. The idea is to improve the spectral properties of the linear systems in the aforementioned analyses by the application of a low-fidelity aerodynamic model as a preconditioner. Subsequently, the linear systems can be solved by fewer number of iterations, efficiently reducing the overall computation time. The solution method is demonstrated on the Onera M6 wing. The structural model of a classical wing box layout is solved by Nastran whereas the aerodynamic pressure distribution is obtained by solving the Euler equations using elsA. The low-fidelity sensitivities used to construct the preconditioner are obtained by an inhouse vortex-lattice code. The multi-fidelity approach shows improved convergence properties especially for flexible wings.

I. Introduction

The trend of employing high-fidelity models in the early design stages has become more pronounced with increasing computational resources, the advent of code parallelism and improving solution algorithms.[1–4] Nevertheless, high-fidelity models based on the Euler or the RANS equations are still considered computationally expensive. This has resulted in efforts to accelerate the convergence of high-fidelity analysis and sensitivity analysis through multi-fidelity modelling.

The idea is to make use of a mathematical model of presumably lower fidelity in order to speed up the convergence of the computationally heavier, but more accurate, models. In this work the high-fidelity aerodynamic model is based on the Euler equations. This enables an estimation of recompression shocks in the transonic flight regime. The low-fidelity aerodynamic model, based on vortex-lattice theory, does not have the predictive capability to capture these nonlinear effects due to the inherent approximations in the governing equations. The structural models in fluid-structure problems are in comparison to the aerodynamic models not considered computationally expensive. A modest number of degrees of freedom is typically sufficient to provide accurate structural deformations and stresses. Consequently, we only apply the multi-fidelity methodology to the aerodynamic models.

II. Methodology

In this work the linear systems in the aeroelastic analysis and sensitivity analysis are solved by the Krylov subspace method, GMRES.[5] Krylov methods, in general, experience faster convergence compared to stationary iterative schemes, such as the Jacobi, the Gauss-Seidel or the SOR method. However, to maintain a robust and efficient convergence rate of GMRES it is necessary to construct a quality preconditioner. The task of the preconditioner is to improve the spectral properties of the system matrix, such that when applied to an iterative solver, the number of iterations decrease. We construct this

*Ph.D. Student, Faculty of Aerospace Engineering, Aerospace Structures and Computational Mechanics, Kluyverweg 1; K.Jovanov@tudelft.nl

†Assistant Professor, Faculty of Aerospace Engineering, Aerospace Structures and Computational Mechanics, Kluyverweg 1; R.deBreuker@tudelft.nl

‡Associate Professor, Faculty of Aerospace Engineering, Aerospace Structures and Computational Mechanics, Kluyverweg 1; M.M.Abdalla@tudelft.nl

§Research Scientist,Aeroelasticity and Structural Dynamics Department, ONERA; Christophe.Blondeau@onera.fr

preconditioner based on the low-fidelity aerodynamic model. Thus we have a physics-based preconditioner that can accelerate the convergence of both the aeroelastic analysis as well as the sensitivity analysis.

III. Results

The results in Figure 1 indicate that the addition of a low-fidelity model significantly reduces the computation time. The test case was the Onera M6 wing at a transonic flight condition of Mach 0.84. The blue line is the covergence rate of the preconditioned system without the low-fidelity model whereas the red line is the conergence rate of the preconditioned system with the low-fidelity model. The increase in efficiency is a factor of 2 for this particular test case.

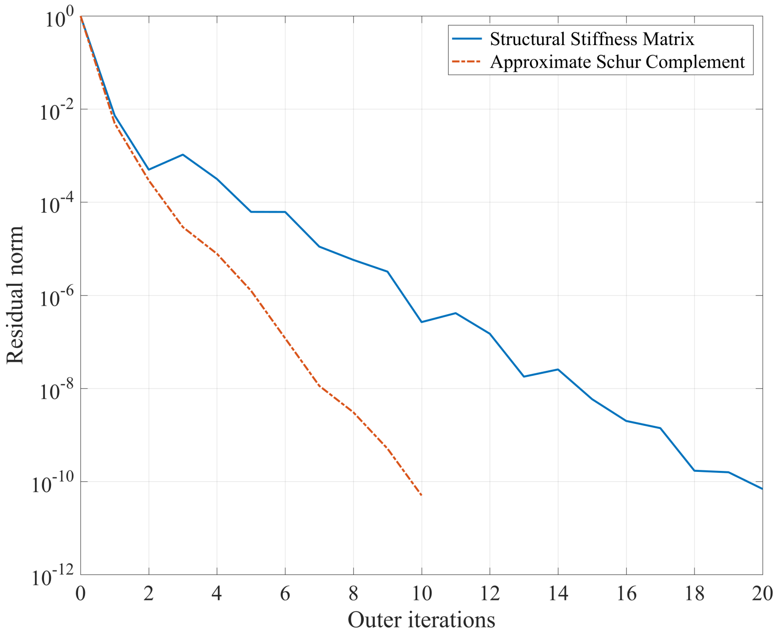


Figure 1: Comparison of convergence behaviour at Mach 0.84

References

[1] Barcelos, M., Bavestrello, H., and Maute, K., "A SchurNewtonKrylov solver for steady-state aeroelastic analysis and design sensitivity analysis," *Computer Methods in Applied Mechanics and Engineering*, Vol. 195, No. 17-18, March 2006, pp. 2050–2069.

[2] Kennedy, G. J. and Martins, J., "Parallel solution methods for aerostructural analysis and design optimization," *Proceedings of the 13th AIAA/ISSMO Multidisciplinary Analysis Optimization Conference, Fort Worth, TX*, 2010.

[3] Kenway, G. K. W., Kennedy, G. J., and Martins, J. R. R. A., "Scalable Parallel Approach for High-Fidelity Steady-State Aeroelastic Analysis and Adjoint Derivative Computations," *AIAA Journal*, Vol. 52, No. 5, May 2014, pp. 935–951.

[4] Blondeau, C., Achard, T., Girodroux-Lavigne, P., and Ohayon, R., "RECENT ACHIEVEMENTS TOWARDS AERO-STRUCTURE GRADIENT COMPUTATION USING HIGH-FIDELITY CFD-CSM IN THE ONERA ELSA SOFTWARE," *International Forum on Aeroelasticity and Structural Dynamics*, 2015.

[5] Saad, Y. and Schulz, H. M., "GMRES: A Generalized Minimal Residual Algorithm For Solving Nonsymmetric Linear Systems," *SIAM J. ScI. STAT. COMPUT.*, Vol. 7, No. 3, July 1986, pp. 856–869.

Incorporation of Bird Strike Requirements in MDO of an Aircraft Wing using Sub-space Metamodels

Jonathan Ollar[1], Royston Jones[2],
Altair Engineering Ltd, Leamington Spa, CV32 4JG, United Kingdom

Vassili Toropov[3]
Queen Mary University of London, London, E1 4NS, United Kingdom

A metamodel based multidisciplinary design optimisation of a conceptual aircraft wing model is presented. The disciplines considered are bird impact at a number of critical locations along the leading edge as well as static bending and twisting stiffness of the wing. The bird strike simulations are many times more costly in terms of computational budget than the the static load cases and as 100 sizing design variables are considered the problem may become very expensive. The multidisciplinary design optimisation is carried out using a method previously proposed by the authors for taking into account disparity in design variable dependence of the disciplines. This design variable dependence is specified by the designer and used to build metamodels in only the space of the significant variables to each discipline. This means that the number of required points for each metamodel, and the associated computational cost for their evaluation, can be reduced. The method is implemented within the optimisation framework known as the mid-range approximation method together with a recovery mechanism for erroneous identification of significant variables. It is shown that, by using the proposed approach to take into account the local design variable dependence of the individual bird strike simulations, the optimisation can be carried out to a much reduced computational budget to what would otherwise be required.

1. Introduction

This paper presents an efficient method of incorporating bird strike as well as stiffness requirements in multidisciplinary optimisation of an aircraft wing including 100 sizing design variables. Bird strike simulations are typically several times more costly than stiffness simulations and as the bird can potentially impact the wing at any location along the leading edge, one have to consider several simulations of the bird impacting critical locations in the same optimisation. Furthermore, gradients of the response functions are not available. This makes the computational cost of the bird strike requirements in the multidisciplinary optimisation problem disproportionately large compared to the stiffness requirements.

In this work, advantage is taken of the fact that each bird impact is a local event, influencing only a small part of the wing, and hence only a small number of the design variables. A method, previously proposed by the authors [1–3], for making use of disparate design variable dependence of the individual disciplines in multidisciplinary optimisation is here used to reduce the number of required evaluations, and hence the overall computational budget of the optimisation. Meta-models are built considering only a subset, of the full set of design variables, significant to the individual disciplines. The method relies on the designer to identify the significant variables for each load case through, for instance, engineering judgement or initial ranking studies. However, if such identification is erroneous a recovery mechanism, implemented as part of a trust-region strategy, is used to recover from resulting metamodelling errors by updating the values of the insignificant variables to align with the current best point at the end of each iteration.

[1]Research Engineer, Altair ProductDesign & PhD candidate, Queen Mary University of London.
[2]Exec VP European Operations & Global CTO, Altair ProductDesign.
[3]Professor of Aerospace Engineering, School of Engineering and Materials Science.

2. Mid-range approximation method

The mid-range approximation method (MAM), also known as the multi-point approximation method, was originally reported by [4, 5] and [6]. The MAM solves a typical constrained optimisation problem in the form:

$$\begin{array}{lll} \underset{\mathbf{x}}{\text{minimize}} & f_0(\mathbf{x}) & \\ \text{subject to} & f_j(\mathbf{x}) \leq 1, & j = 1, \ldots, m \\ & A_i \leq x_i \leq B_i, & i = 1, \ldots, n \end{array} \tag{1}$$

where $f_0(\mathbf{x})$ is the objective function, $f_j(\mathbf{x})$ is the j-th constraint, $\mathbf{x}$ is the vector of design variables and A_i and B_i are the upper and lower bounds respectively on the design variable x_i. The optimisation problem (1) is replaced by a sequence of approximate sub-problems defined as:

$$\begin{array}{lll} \underset{\mathbf{x}}{\text{minimize}} & \tilde{f}_0^{\,k}(\mathbf{x}) & \\ \text{subject to} & \tilde{f}_j^{\,k}(\mathbf{x}) \leq 1, & j = 1, \ldots, m \\ & \left.\begin{array}{l} A_i^k \leq x_i \leq B_i^k \\ A_i^k \geq A_i \\ B_i^k \leq B_i \end{array}\right\} & i = 1, \ldots, n \end{array} \tag{2}$$

where k denotes the current iteration number. $\tilde{f}_0^{\,k}(\mathbf{x})$ is a metamodel of the objective function and $\tilde{f}_j^{\,k}(\mathbf{x})$ is a metamodel of the j-th constraint function, both considered valid only in the current trust region. A_i^k and B_i^k are the bounds of the current trust region where the sub-problem (2) is solved for the current iteration. The solution procedure for each sub-problem consists of sampling, creating metamodels, solving the approximate optimisation problem and determining a new location and size of the trust region for the next iteration. The trust region will move and change size after each iteration until the termination criterion is reached. Figure 1 illustrates the history of trust regions through the sequence of sub problems in two dimensions. The trust region strategy has gone through several developments to account for the presence of numerical noise in the response function values [7, 8], occasional simulation failures [9], and improvements for high performance computing [10]. In this work a doe technique based on extensible lattice sequences [11], and a kriging metamodelling technique as outlined in [12], is used.

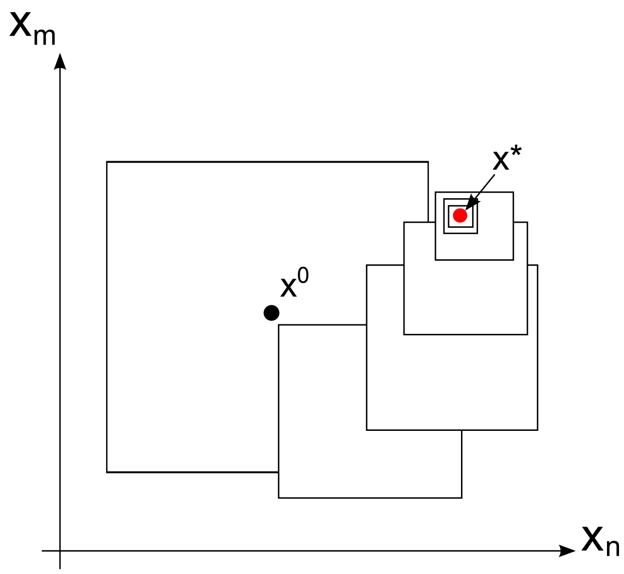


Figure 1: Typical history of the trust regions. In every iteration of the optimization process the new trust region is centered around the current solution and either kept the same size, reduced or enlarged.

Wingbox adaptive parametric modeling and its application to structural optimization

Anna L. Arsenyeva* and Fabian Duddeck,†
Chair of Computational Mechanics, Technische Universität München (TUM), Munich, 80333, Germany

I. Introduction

In current research development of parametric wing model and its application to optimization process are considered. Due to high computational cost of the wing structural optimization problem when aerodynamics, structural analysis and dynamic analysis should be considered, makes optimization very expensive. Another problem is possible discontinuities of the optimization problem, due to present integer parameters, e.g. number of ribs/stringers. This research is aimed to address this issues by introducing flexible and fast parameterized wingbox model and using two-stage optimization approach, which combines Evolutionary Algorithms (EA) and local search algorithms.

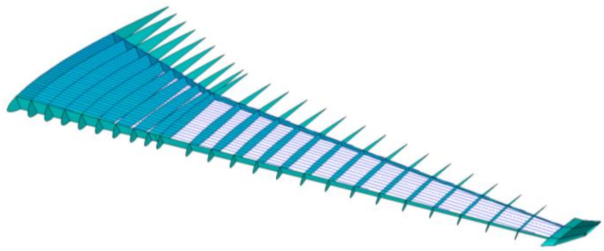


Figure 1. Wingbox components definition for wing

The wing aerodynamic shape is defined by a set of parameterized NACA airfoils. In general, any closed wing skin surface can be imported into the process, due to the adaptive way of the internal structure definition. Different parameters of internal wing structure can be varied, e.g. number and location of ribs/spars/stringers, their shape, the thicknesses, such as the linear varying of the thickness along the length for skin and spars components. Geometrical and FE models of the wing are generated using Python and ANSYS APDL scripting. In Fig.1 the internal components of wing model are shown, which are used in the current optimization setup.

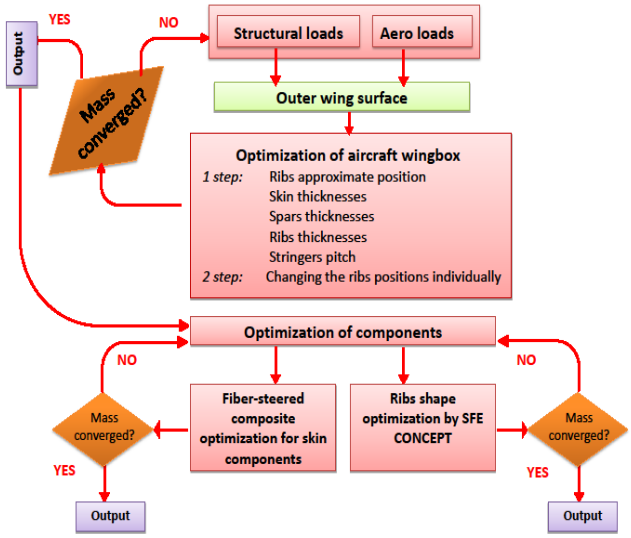


Figure 2. Framework for optimization of wing components

Several types of loads are considered, including acceleration load, engine loads and aero loads. The aero loads can be calculated using ANSYS Fluent CFD or simplified XFOIL 2D code. The flow simulation is calculated for a fixed external shape and obtained pressure distribution is mapped automatically to existing structural model in ANSYS Mechanical.

The two level optimization process is proposed as shown in Fig.2. The first level involves optimization of the "global" parameters of wingbox such as number, position and approximate thicknesses for various components. The objective here is to reduce the mass, while satisfying local skin deflection and buckling constraints. Second level includes optimization of the wingbox subcomponents (ribs, spars, etc.) for fixed global layout of the wingbox, obtained in first level. Subcomponent optimization can include different approaches such as the steered fiber composite optimization for the wing skin [3], optimization of ribs topology and shape [4]. In the current paper only the first optimization level is considered.

At the beginning, the optimal number and positions of the ribs will be determined. In order to reduce the number of wingbox parameters and consequently the computational effort for the optimization, two different parameterization levels are introduced. At the first step, the coarse parameterization is used, when the rib

---

* Research assistant, Chair of Computational Mechanics, anna.arsenyeva@tum.de.
† Prof. Dr., Chair of Computational Mechanics, duddeck@tum.de.

spacing for each section is defined using the pitch parameter, which can vary linearly within every section. Thus, two parameters define rib positions for each section (pitch and pitch gradient), resulting in total 4 parameters. Low number of design parameters in combination with fast and flexible wingbox model allow to employ global stochastic optimization algorithms, e.g. EA, which can deal with non-smooth/non-continuous dependencies, which are present in this problem due to the changing number of ribs/stringers.

At second stage finer parameterization is applied, when the position and angle of each rib can be varied individually in the vicinity of the previously obtained solution by moving the rib's start and end points along the spars. Global optimal solution from the first step defines the initial ribs positions for the refined optimization. At this step, local optimization methods (e.g. gradient-based methods, COBYLA) are used to obtain refined design.

I. Sensitivity analysis.

The impact of used design variables (in total 14 parameters) on the structural responses such as weight, maximum local skin deflection between each pair of ribs and maximum critical buckling load is determined for different aero loads computation methods. Two numerical experiments, generated using Latin-Hypercube sampling within the coarse parameters, are performed for the models with XFOIL and ANSYS FLUENT. The resulting Spearman coefficient matrices for correlations between responses and the coarse design parameters are shown in Fig. 3.

As can be seen XFOIL results gives the very similar correlations, compared to the results obtained for Fluent, meaning that it can at least capture general trends accurately. Considering that XFOIL approach is much faster, compared to full CFD analysis, it can be used for the preliminary global optimization studies.

In Fig. 4 shown the exemplary optimal results obtained using two step approach. In this optimization run only ribs locations where optimized, which are defined by four parameters at the course level and two times the number of ribs parameters at the fine level. After the EA optimization with approx. 2800 evaluations, the best found feasible design (see Fig left) has the overall mass of 2907.4 kg. The refinement step with only 230 evaluations further reduces the mass 2828 kg (see Fig. right).

In the current research the adaptive wing model is developed and tested, the first optimal results were obtained. In the further research, the wingbox internal structure will be optimized with the use of implemented model with more realistic outer wing dimensions and shape (similar to the A320 wing).

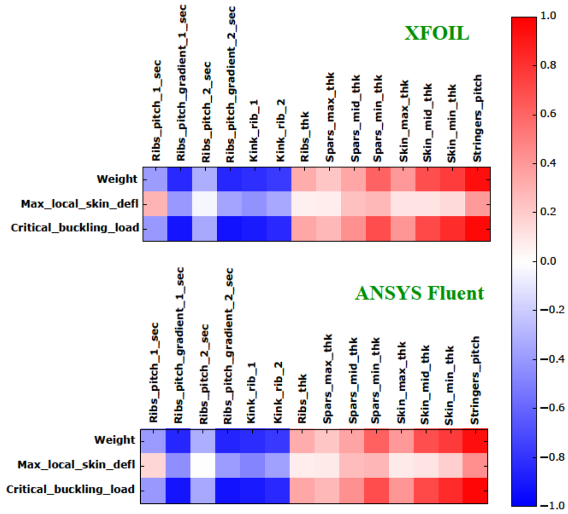


Figure 3. The resulting Spearman coefficient matrices: 300 designs

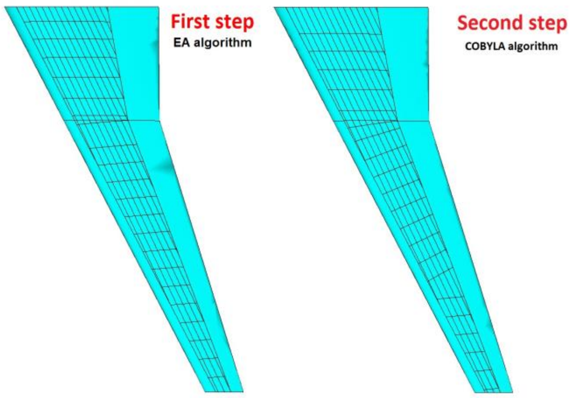


Figure 4. EA results, coarse model, COBYLA refined optimization

References

[1] F. Dorbath. “A Flexible Wing Modeling and Physical Mass Estimation System for Early Aircraft Design Stages”, PhD thesis, TU at Hamburg-Harburg, 2014

[2] M. Chun-Yung Niu, “Airframe-structural design. Practical design information and data on Aircraft structures”, Conmilit Press Ltd, 1989

[3] S.H. Hesse, A. Arsenyeva, D.H.-J.A. Lukaszewicz, F. Duddeck, “A Parameterized Surface Method to Determine Optimal Variable Stiffness Layup by Global Search”, In: 20th International Conference on Composite Materials, Copenhagen, Denmark, 2015

[4] A. Arsenyeva, F. Duddeck, “Efficient and adaptive parametric modeling for shape optimization of a wingbox”, In: 3rd ECCOMAS Young Investigators Conference, Aachen, Germany, 2015

Improved adjoint solver for complex flow conditions

Shenren Xu[*] and Sebastian Timme[†]
University of Liverpool, Liverpool L69 3GH, United Kingdom

In this work, a deflated Krylov subspace linear solver, GCRO-DR, preconditioned with either incomplete LU factorisation or multigrid, is used to solve the adjoint equation. The DLR–TAU code is used in this work.

I. Introduction to design under complex flow conditions

The essential step in performing gradient-based shape optimisation using the Reynolds-averaged Navier–Stokes equations is to compute the adjoint solution. For complex flow conditions such as flows exhibiting large separation, the resulting adjoint equation could be extremely stiff to solve, while shape optimization is as interesting under these conditions as for design condition.

II. Deflated Krylov methods for solving the adjoint equation

In this work, a deflated Krylov subspace linear solver, GCRO-DR, preconditioned with either incomplete LU factorisation or multigrid, is used to solve the adjoint equation. The DLR–TAU code is used in this work.

III. Results for DLR F6 at both cruise and off-design conditions

The steady-state flow solutions for the DLR F6 transonic turbulent case are shown on the left in Figs. 1 and 2 where the stationary separation bubbles at the wing-body junction towards the wing trailing edge are clearly visible. The convergence history of the adjoint equations for both flow conditions using GMRES and GCRO-DR are shown on the right in Figs. 1 and 2. All adjoint solves are performed on 48 cores. For cruise condition, GCRO-DR moderately speeds up the convergence with less memory. The speedup and memory reduction is much more significant for the more complex flow at 4.5 deg angle of attack.

---

[*]Research associate, School of Engineering.
[†]Lecturer, School of Engineering.

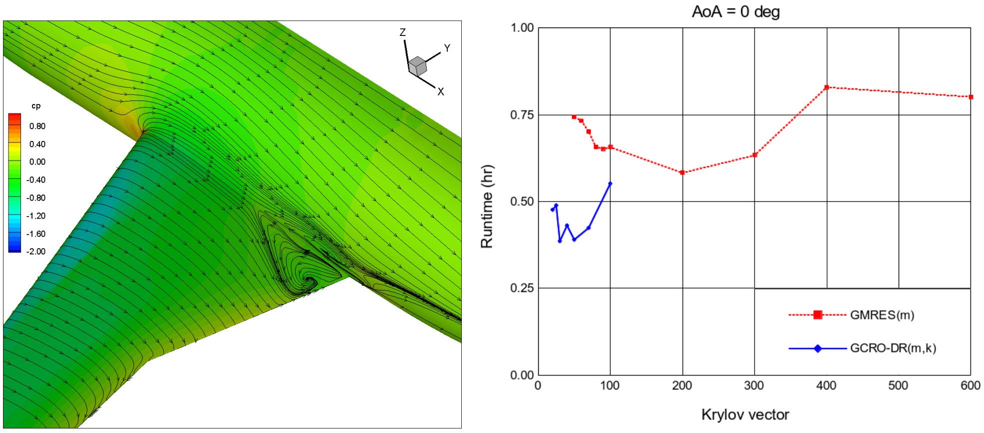


Figure 1. Left: flow solution at zero deg angle of attack. Skin friction lines plotted based on the surface shear force vector field. Right: Runtime of both linear solver plotted against the number of Krylov vectors used.

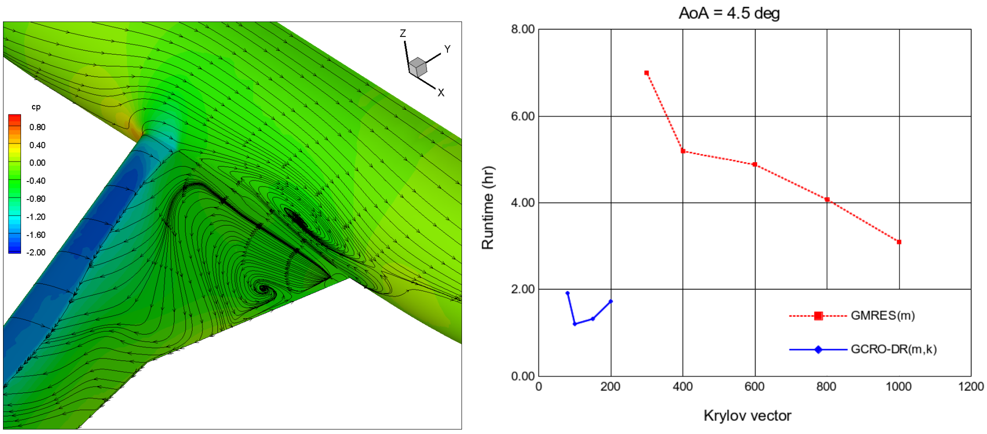


Figure 2. Left: flow solution at 4.5 deg angle of attack. Skin friction lines plotted based on the surface shear force vector field. Right: Runtime of both linear solver plotted against the number of Krylov vectors used.

Adjoint-based Optimization Techniques in SU2 with Applications to Industrial Flows

Thomas D. Economon*

Stanford University, Stanford, CA, 94305, USA.

More than 25 years after the landmark article by Prof. Antony Jameson on the topic,[1] adjoint-based design and grid adaptation techniques can be found in many CFD solvers and have been successfully demonstrated across a wide range of applications. Recent advances are improving the robustness and accuracy of these methods even for the complex geometries and unstructured grids needed within industrial environments. The objective of this presentation is to overview the adjoint-based methods available within the open-source SU2 software suite and how, along with native modules for geometry parameterization and mesh deformation, the methods can be applied to industry-scale design problems. Shape optimization results for the NASA Common Research Model (CRM) and Lockheed Martin 1021 (LM1021) aircraft geometries will be presented.

I. Introduction

SU2 is an open-source integrated analysis and design tool kit for solving complex, multi-disciplinary problems on unstructured computational grids.[2] At its core, the software suite is a collection of C++ modules, linked within a Python framework, that performs the discretization and solution of partial differential equation-based (PDE) problems and executes PDE-constrained optimization tasks. The object-oriented architecture of SU2 is easy to read, learn, and modify in order to treat unique problems across a wide range of engineering disciplines. These capabilities and the open-source philosophy position SU2 uniquely within the CFD community to become a test-bed for computational scientists interested in the high-fidelity analysis and design of complex engineering systems.

The objective of this presentation is to overview the adjoint-based methods available within the SU2 suite and how, along with native modules for geometry parameterization and mesh deformation, the methods can be applied to industry-scale shape design problems.

II. Adjoint Methods within SU2

Owing to the open-source community, the SU2 code contains two distinctly different implementations of the adjoint methodology that have been completed by experts in the field: a continuous adjoint by Stanford University and a discrete adjoint via algorithmic differentiation (AD) of the entire solver by the Technical University of Kaiserslautern.

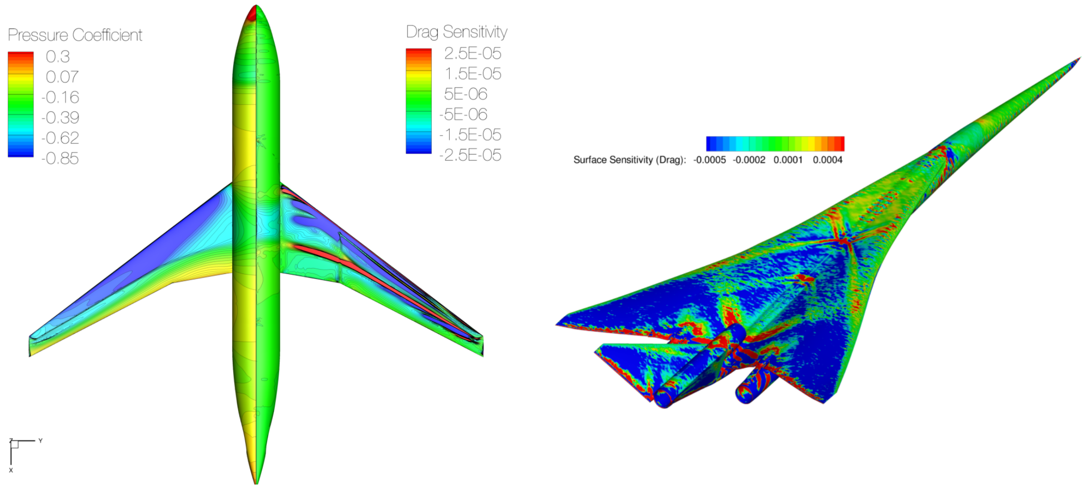
SU2 features a native continuous adjoint solver in a surface formulation based on the frozen turbulent viscosity assumption.[3] Since debuting in the initial public release of SU2, the continuous adjoint solver has been extensively used and rigorously tested[4,5] for both inviscid and viscous problems across many flow regimes.

More recently, a discrete adjoint solver in SU2 has been generated by algorithmically differentiating the entire codebase. To be more precise, this includes not only the flux routines, but also the complete nonlinear iteration structure of the solver. To improve efficiency, the group at the Technical University of Kaiserslautern has developed a custom set of AD tools specifically tuned for CFD applications. Currently, the discrete adjoint solver has been implemented for the Euler, Navier-Stokes, and RANS equations (S-A and SST turbulence models) in combination with all of the existing design parameterizations available in SU2.[6,7]

Modules for performing flow and adjoint solutions, acquiring gradient information by projecting sensitivities into the design space, and mesh deformation techniques are all included in the suite, amongst others. Typically, geometry parameterization is accomplished with a free-form deformation approach,

*Postdoctoral Scholar, Department of Aeronautics & Astronautics, economon@stanford.edu

mesh deformation with an elasticity-based approach, and the shape design loop is driven by the SLSQP optimizer found in the SciPy library.[8]



(a) Pressure coefficient (left) and surface sensitivity (right) contours on the upper surface of the CRM configuration.

(b) Drag sensitivity contours on the LM1021 configuration as computed by SU2.

Figure 1. Two applications of optimal shape design to industrial flow problems with SU2.

III. Shape Design Applications

After demonstrating some ideas on simple test cases, two examples of applying SU2 to shape design of industry-scale problems will be presented: the NASA Common Research Model (CRM) and Lockheed Martin 1021 aircraft geometries, as shown in Fig. 1.

The NASA CRM has been selected as a baseline model for further investigation. It was developed to be used in CFD validation exercises as part of the fourth AIAA CFD Drag Prediction Workshop.[9] This wing-body-tail configuration provides a challenging problem due to its geometric complexity and the large, unstructured meshes employed during its analysis. In terms of the proposed design problem, the objective will be to minimize drag while imposing lift and moment constraints. Geometrical constraints will be included to guarantee the feasibility of the final design.

In recent years, there has been renewed interest in low-boom supersonic aircraft. Advances in simulation-based design, including adjoint methods, are opening the door to new supersonic aircraft designs with reduced sonic boom impacts. The Lockheed Martin 1021 (LM1021) is one of the test cases from the AIAA 1st Sonic Boom Prediction Workshop,[10] and it will be used to demonstrate adjoint-based optimization of a supersonic configuration.

References

[1]A. Jameson. Aerodynamic design via control theory. *Journal of Scientific Computing*, 3(3):233–260, 1988.

[2]T. D. Economon, F. Palacios, S. R. Copeland, T. W. Lukaczyk, and J. J. Alonso. SU2: An open-source suite for multiphysics simulation and design. *AIAA Journal*, 54(3):828–846, 2015.

[3]C. Castro, C. Lozano, F. Palacios, and E. Zuazua. A systematic continuous adjoint approach to viscous aerodynamic design on unstructured grids. *AIAA Journal*, 45(9):2125–2139, 2007. DOI: 10.2514/1.24859.

[4]F. Palacios, T. D. Economon, and J. J. Alonso. Large-scale aircraft design using SU2. AIAA Paper 2015-1946, January 2015.

[5]T. D. Economon, F. Palacios, and J. J. Alonso. Unsteady continuous adjoint approach for aerodynamic design on dynamic meshes. *AIAA Journal*, 53(9):2437–2453, 2015.

[6]T. Albring, M. Sagebaum, and N.R. Gauger. Development of a consistent discrete adjoint solver in an evolving aerodynamic design framework. AIAA Paper 2015-3240, 2015.

[7]B.Y. Zhou, T. Albring, N.R. Gauger, T. D. Economon, F. Palacios, and J. J. Alonso. A discrete adjoint framework for unsteady aerodynamic and aeroacoustic optimizatio. AIAA Paper 2015-3355, 2015.

[8]E. Jones, T. Oliphant, P. Peterson, et al. SciPy: Open source scientific tools for Python, 2001.

[9]J. C. Vassberg, M. A. DeHaan, S. M. Rivers, and R. A. Wahls. Development of a common research model for applied CFD validation studies. AIAA Paper 2008-6919, 2008.

[10]J. M. Morgenstern, M. Buonanno, and F. Marconi. Full configuration low boom model and grids for 2014 sonic boom prediction workshop. Number AIAA Paper 2013-647, 2013.

Introducing STAMPS: an open-source discrete adjoint CFD solver using source-transformation AD

Jens-Dominik Müller[*]
and Mateusz Gugala[†], Shenren Xu[‡], Jan Hückelheim[§]
, Pavanakumar Mohanamuraly[¶], Oluwadamilare Rahman Imam-Lawal[‖]
Queen Mary University of London, London, E1 4NS, United Kingdom

We present the developments of the open-source adjoint CFD code STAMPS which uses source-transformation AD and fixed-point treatments to obtain readable, modifiable and efficient adjoint code.

I. Introduction

Adjoint CFD solvers are now commonplace in the aerospace industry, effectively all major aerospace companies and research labs develop or collaborate around in-house implementations. Similarly, the major code vendors now offer adjoint variants with varying capability. There are also two major adjoint open-source efforts, the incompressible OpenFOAM[1] and the compressible SU2.[2] Both codes use the continuous approach as the main adjoint variant, but also have discrete variants using operator overloading AD tools.[3,4]

The continuous variants can produce effective adjoint code for the robust time-stepping of the compressible flow equations, and can be made sufficiently stable for pressure-correction type discretisations. As long as the user only needs a RANS solution for the flow, adjoint solutions can be obtained for large-scale problems. For modelling beyond RANS, the continuous approach suffers from difficulties with stability, as well as from a substantial manual effort in deriving and discretising the adjoint equations for updates and new physical models. Questions remain whether the fact that the computed gradients are not the gradients of the discrete CFD solutions pose a problem for applications of continuous adjoints in e.g. uncertainty quantification.

The discrete adjoints derived with automatic differentiation tools can in principle overcome these difficulties: exact transposition in reverse-mode differentiation guarantees that the linear stability of the CFD code (primal) is inherited, any code modifications of the primal are automatically differentiated and directly available to the user.

However, there are major caveats. Firstly, the initial effort of amending the primal to make AD successful is often substantial. Good results typically need close collaboration of the teams developing the CFD solver with the AD tool developers. Secondly, stability does remain an issue. In the memory-effective AD implementations based on a steady-state primal, only linear stability is inherited. Stability due to non-linear switching of the stabilisation as e.g. in limit cycle oscillations (LCO) is lost when the system Jacobian is frozen. Thirdly, and most importantly, for advanced object-oriented languages such as C++ AD tools using the source-transformation (S-T) approach are not available. Instead operator-overloading (O-O) approaches have to be used which record a 'tape' of the relevant operations of the primal and unwinds that backward for the adjoint. Even with shortcuts such as not differentiating through linear solvers, the memory requirements are substantial and not appropriate for industrial or large-scale computation on typically available memory.

The CFD optimisation group at QMUL has over the years developed the discrete adjoint solver STAMPS[a] using automatic differentiation. Different from other groups we have not succumbed to the

[*]Senior lecturer, School for Engineering and Materials Science, Queen Mary University of London, j.mueller@qmul.ac.uk
[†]PhD candidate, SEMS, QMUL, m.gugala@qmul.ac.uk
[‡]Research Associate, current address: School of Engineering, Liverpool University, xushenren@gmail.com
[§]PhD candidate, SEMS, QMUL, j.c.hueckelheim@qmul.ac.uk
[¶]PhD candidate, SEMS, QMUL, p.mohanamuraly@qmul.ac.uk
[‖]PhD candidate, SEMS, QMUL, o.r.imam-lawal@qmul.ac.uk
[a]Source-Transformation Adjoint Multi-Physics Solver

temptation of C++, but use the level of object-orientedness available in Fortran90. In turn this allows us to use the source-transformation AD-tool Tapenade, and hence avoid the huge memory requirements of O-O and produce AD-derived adjoint code that is as efficient has hand-differentiated code.[5,6] To date, there is no AD-derived adjoint using S-T. This in our view holds back adoption of adjoint methods for applications other than "shape improvement", multi-physics or efficient uncertainty quantification in the wider scientific community. In the Autumn 2016 we will therefore make the STAMPS code open-source.

II. Current status of development of STAMPS

The key functionality of STAMPS covers:

- Edge-based compressible finite-volume discretisation using MUSCL reconstruction on hybrid elements with geometric multigrid,[7]
- Explicit, Block-Jacobi and implicit JT-KIRK timestepping,[8]
- Parallelisation on CPU and GPU with OpenMP, MPI and the hybrid OP2 library,[9]
- Inviscid, laminar and turbulent flow calculation using RANS, DES and LES,
- Discrete adjoint solver automatically built from the primal using source-transformation AD with efficient treatment of fixed-point iterations and reverse-differentiated shape sensitivity calculation,
- Laplacian smoothing and linear elasticity mesh deformation algorithms,
- Mesh r-refinement with truncation error and output error estimates.

Among the unique solver features of the flow solver one can distinguish a robust and stable JT-KIRK[8] solver with geometric multi-grid and CFL auto-ramping technique[10] used to obtain a stable yet fast convergence path. The use of strong preconditioning with a first-order Jacobin approximation allows to extend stability of the solver as compared to the explicit or simple block Jacobian scheme and obtain steady state for cases with small or mild unsteadiness.

We use the Tapenade source-to-source transformation AD tool[11] to generate derivative code of selected routines. The main advantage of using Tapenade is that its generated derivative code has a low memory footprint. The adjoint solver in STAMPS is hand assembled using the generated derivative code to avoid differentiation of the fixed point iteration loop that is used to solve the primal system of equations. The MPI support is an ongoing development.

The presentation will present the design and capabilities of the code, including validation and performance analysis against standard testcases and application examples of adjoint optimisation. A roadmap for publication and future development will also be shown.

References

[1]Othmer, C., "A continuous adjoint formulation for the computation of topological and surface sensitivities of ducted flows," *International Journal for Numerical Methods in Fluids*, Vol. 58, No. 8, 2008, pp. 861–877, DOI 10.1002/fld.1770.

[2]Palacios, F., Colonno, M. R., Aranake, A. C., Campos, A., Copeland, S. R., Economon, T. D., Lonkar, A. K., Lukaczyk, T. W., Taylor, T. W., and Alonso, J. J., "Stanford University Unstructured (SU2): An open-source integrated computational environment for multi-physics simulation and design," *AIAA Paper*, Vol. 287, 2013, pp. 2013.

[3]Towara, M. and Naumann, U., "A discrete adjoint model for OpenFOAM," *Procedia Computer Science*, Vol. 18, 2013, pp. 429–438.

[4]Sagebaum, M., Gauger, N. R., Naumann, U., Lotz, J., and Leppkes, K., "Algorithmic differentiation of a complex C++ code with underlying libraries," *Procedia Computer Science*, Vol. 18, 2013, pp. 208–217.

[5]Cusdin, P. and Müller, J.-D., "On the Performance of discrete adjoint CFD codes using automatic differentiation," *International Journal for Numerical Methods in Fluids*, Vol. 47, No. 6-7, 2005, pp. 939–945.

[6]Christakopoulos, F., Jones, D., and Müller, J.-D., "Pseudo-timestepping and validation for Automatic Differentiation derived code," *Computers and Fluids*, Vol. 46, No. 1, 2011, pp. 174–179.

[7]Moinier, P., Müller, J.-D., and Giles, M. B., "Edge-Based Multigrid Schemes and Preconditioning for Hybrid Grids," *AIAA Journal*, Vol. 40, No. 10, 2002, pp. 1954–60.

[8]Xu, S., Radford, D., Meyer, M., and Müller, J.-D., "Stabilisation of discrete steady adjoint solvers," *Journal of Computational Physics*, Vol. 299, 2015, pp. 175–195.

[9]Giles, M. B., Mudalige, G. R., Sharif, Z., Markall, G., and Kelly, P. H., "Performance analysis and optimization of the OP2 framework on many-core architectures," *The Computer Journal*, 2011, pp. bxr062.

[10]Michalak, C. and Ollivier-Gooch, C., "Globalized matrix-explicit Newton-GMRES for the high-order accurate solution of the Euler equations," *Computers & Fluids*, Vol. 39, No. 7, 2010, pp. 1156–1167.

[11]Hascoët, L. and Pascual, V., "The Tapenade Automatic Differentiation tool: Principles, Model, and Specification," *ACM Transactions On Mathematical Software*, Vol. 39, No. 3, 2013.

Effective discrete adjoint OpenFOAM for volume and surface sensitivities

Arindam Sen* and Markus Towara* and Uwe Naumann†

Software and Tools for Computational Engineering, RWTH Aachen University, 52074, Germany

A discrete adjoint version of OpenFOAM is presented here. This version can be used to accurately compute sensitivities, i.e derivatives of suitable CFD objective functions such as drag, lift, pressure loss etc. with respect to desired design variables such as porosity or mesh points which can then be utilized for the purpose of gradient based design optimization.

I. Introduction

Adjoint methods for gradient based optimization are now widely used in CFD applications. Open source CFD tools such as OpenFOAM are used by a vast user base in both industry and academia for various applications. The objective of this work is to develop a discrete adjoint version of OpenFoam using Algorithmic Differentiation[1] that helps to compute efficient and accurate (up to machine precision) sensitivities, i.e. gradients of specific objective functions such as drag, lift, pressure loss etc. with respect to desired design variables[2].[3] This discrete adjoint version is obtained by overloading all the basic mathematical operations using a custom data type with the aid of operator overloading tool dco/c++.[4] Compared to continuous adjoints, this framework is flexible and easily adaptable. Also, because of the highly objected-oriented nature of the source code (in C++), source transformation tools like Tapenade[5] are not suitable. A black box adjoint usually incurs an unaffordable memory footprint. To overcome this limitation, several standard improvement techniques like binomial checkpointing,[6] symbolic differentiation of the embedded linear iterative solver,[7] reverse accumulation for steady state problems[8] were implemented. This framework is also extended to compute higher order adjoints. A discrete adjoint version of Foam-extend, a fork of OpenFOAM with additional functionalities, was also obtained. For incompressible steady-state problems, the adjoint of the coupled implicit solver was used to gain further performance improvement.[9] This framework was then tested on small to medium scale problems for a range of applications like ducted flows, external aerodynamics and conjugate heat transfer with qualitative validation against the continuous adjoint implementation.[10] The performance, results and challenges are subsequently discussed.

II. Discrete adjoint OpenFOAM

The optimization problem can be represented as $J : I\!R^n \rightarrow I\!R^m$, $J(\alpha) \rightarrow min!$, where n denotes the number of input variables (design space) and m denotes the number of output variables (dimension of the objective function). For the purpose of topology or shape optimization, $m = 1$ or $O(1)$ and $n \gg m$. For such a problem, the cost of obtaining the derivative, $\frac{\partial J}{\partial \alpha}$ using finite differences is $n * Cost(J)$ whereas that using adjoint is $m * Cost(J)$, thus becoming the obvious method of preference.

The basic method for obtaining a black box adjoint of a decoupled incompressible solver in OpenFOAM called *simpleFoam* using dco/c++ has previously been discussed.[2] This framework has then been extented to an unsteady solver and a fully-coupled implicit solver based on Foam-extend[3].[9]

A black box adjoint has limited usefulness in the context of relevant CFD problems pertaining to high memory footprint. Extension of this framework has been achieved by application of standard improvement techniques such as checkpointing and symbolic differentiation of the embedded linear iterative solver to obtain significant performance improvement[3].[9] Additionally for steady state problems, the application of reverse accumulation allows us to adjoin the last non-linear iteration step repeatedly by changing the inputs once the forward evaluations have converged contractively to a fixed point. This method allows

*PhD Candidate, Software and Tools for Computational Engineering, RWTH Aachen University, Germany

†Professor, Software and Tools for Computational Engineering, RWTH Aachen University, Germany

the adjoint solution to usually converge faster and does not need additional memory for storing the checkpoints.

This framework is consistently applicable for a variation of flow characteristics like laminar/turbulent, steady/unsteady, compressible/incompressible and hence has a major advantage over it's continuous counterpart which typically involves tedious mathematical derivations for different flow physics.

III. Results

The framework described above is tested on small to medium scale CFD applications. Fig 1. shows the volume sensitivities of an airduct design domain (Courtesy: Volkswagen AG), i.e derivatives of pressure loss between the inlet and outlet with respect to the porosity term α. The design domain consists of 5 Mio. volume cells and the primal solution is obtained using *simpleFoam* for a Reynolds number 10. Fig. 2 shows the surface sensitivities of a 3D Onera M6, i.e derivatives of drag force on wing with respect to mesh points at a Mach number of 0.84.

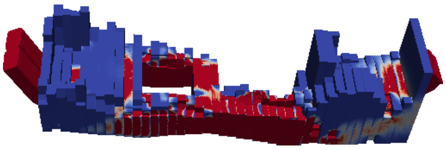


Figure 1. Positive volume sensitivities of airduct design domain. Blue color depicting region where optimization could potentially remove material

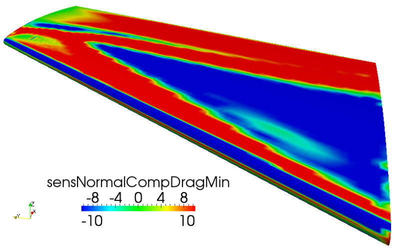


Figure 2. Surface sensitivities of ONERA M6. For drag minimization, Red color: inward displacement, Blue Color: outward displacement

References

[1]U. Naumann. The Art of Differentiating Computer Programs. An Introduction to Algorithmic Differentiation. , Chapter 1 and 2. SIAM, 2012

[2]M.Towara and U.Naumann. A discrete adjoint model for OpenFOAM. Procedia Computer Science , 18(0):429 438, 2013. 2013 International Conference on Computational Science

[3]A.Sen, M.Towara, and U.Naumann. A discrete adjoint version of an unsteady incompressible solver for OpenFOAM using algorithmic differentiation. WCCM XI ECCM V ECFD VI, Barcelona, 2014

[4]J. Lotz, K. Leppkes, and U. Naumann. dco/c++ -derivative code by overloading in C++. Aachener Informatik Berichte (AIB-2011-06), 2011

[5]L. Hascot and V. Pascual. The Tapenade Automatic Differentiation tool: Principles, Model, and Specification. ACM Transactions On Mathematical Software , 39(3), 2013

[6]A. Griewank and A. Walther. Algorithm 799: Revolve: An implementation of checkpointing for the reverse or adjoint mode of computational differentiation. ACM Transactions on Mathematical Software , 26(1):1945, March 2000.

[7]M. B. Giles. Collected matrix derivative results for forward and reverse mode algorithmic differentiation. In Advances in Automatic Differentiation , pages 3544. Springer, 2008

[8]B.Christianson. Reverse accumulation and attractive fixed points. Optimization methods and software, Vol. 3, No. 4, 1994, p. 311-326

[9]A. Sen, M. Towara, and U. Naumann, Sensitivity computation for ducted flows using adjoint of implicit pressure-velocity coupled solver based on Foam, EUROGEN 2015, 14-16 September 2015, Glasgow, UK .

[10]C. Othmer,E. de Villiers and H.G. Weller, Implementation of a continuous adjoint for topology optimization of ducted flows, AIAAA, Miami, 2007.

Output-based r-refinement using a flow-coupled system solve.

Mateusz Gugala* and Jens-Dominik Müller†

Queen Mary University of London, London, E1 4NS, United Kingdom

I. Introduction

The adjoint method[1] is well-established as the most efficient method for aerodynamic shape optimisation with CFD. The adjoint solution quantifies the linearised effect of a unit source term in the conservation equation in each mesh point. If the local error is expressed as a source term, then the adjoint can be also used to drive a solution-adaptive mesh refinement process that computes a more accurate objective function at lower computational cost compared to error-estimators without adjoint weighting or compared to heuristic sensors. The adjoint-weighted truncation error known also as an output-based indicator gained popularity as an effective driver for an adaptation process - see e.g.[2–4] Through the adjoint weighting, the mesh adaptation is very effectively targeted to those areas of the computational domain where the objective function is highly sensitive to local error. This is the key advantage of an output-based indicator as compared to other approaches such as e.g. gradient/Hessian-based sensors or unweighted truncation-error-based sensors. While the latter at least attempts to estimate the actual errors, both of these methods apply the refinement to all errors, regardless of whether they are relevant to the computation of the objective function or not.

The adjoint-weighted adaptation sensor can be used as a driver for the refinement process in various ways, among which the three main techniques are: h-refinement, $r-$refinement and p-refinement, in this work we focus on $r-$refinement, of which several examples can be found in the literature. Dwight[5] proposed using linear elasticity mesh deformation in order to cluster the nodes in the areas with highest sensor values, which can be achieved by applying a non-zero body force term. Tyson et al.[6] compared several methods based on the fundamental principle of equi-distributing of a weight function across the domain where the weight function is some measure of the local error. However, these methods do not connect the system of flow equations and their discretisation to the deformation methodology which can result in non-optimal distribution of computational points for a general case.

In this work, we present an alternative methodology for $r-$refinement which couples deformation algorithm with the system of flow equations, with the key goal to obtain an optimal node distribution and mesh quality for the given error estimate, discretisation scheme, and flow solution. The cube case with a 3D manufactured solution is used for testing purposes and later the technique is applied to the Onera M6 wing case.

II. A flow-coupled r-refinement methodology

The output error (OE) provides the information on how the truncation errors (TE) in each control volume and for each equation contribute to the error in the output of interest e.g. lift or drag. More formally it can be derived as the adjoint-weighted truncation error presented in equation (1). The subscript h stands for the discrete space with characteristic mesh size h.

$$OE_h = v_h^T TE_h. \tag{1}$$

The truncation error estimate i.e. TE_h is obtained using a multi-grid approach between a fine mesh h and a topologically inconsistent coarse mesh H

$$TE_h = -\mathcal{I}_H^h R_H(\mathcal{I}_h^H U_h), \tag{2}$$

*PhD candidate, School for Engineering and Materials Science, m.gugala@qmul.ac.uk

†Senior lecturer, School for Engineering and Materials Science, j.mueller@qmul.ac.uk

where the operator $\mathcal{I}$ stands for inter-grid interpolation. The complete methodology and derivations are presented by the author in.[7]

For the system of flow equations $R_h(U_h, X_h)$ the Jacobian matrix $\left(\frac{\partial R}{\partial X}\right)_h$ relates the truncation error with the nodal positions X, thus when displacing the nodes in the negative gradient direction the truncation errors will decrease. The deformation algorithm based on the least squares minimisation problem formed using the output error (OE_h) as a driving term, and the Jacobian $\left(\frac{\partial R}{\partial X}\right)_h$ as a system matrix is derived as presented in Eq. (3).

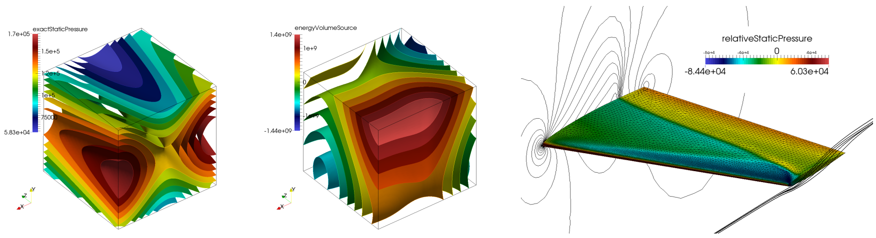
$$\min_{x} \left|\left| \left(\frac{\partial R}{\partial X}\right)_h \delta X_h - OE_h \right|\right|_2 \tag{3}$$

The Jacobian $\frac{\partial R}{\partial X}$ is derived using the AD tool Tapenade[8] applied to the discretised system $R_h(U_h, X_h)$ with the vector X_h as an independent variable. As a result of strong coupling of the formulated least square system (3) with discretised flow equations, the information on the error directionality, not only magnitude should be obtained, thus leading to optimal nodes displacement.

The in-house flow and adjoint solver mgopt developed at Queen Mary University of London is used in this work. It is a finite-volume, compressible, vertex-centred code with an edge-based data structure.

III. Results

The cube case with a 3-D manufactured solution by Roy[9] is used for verification of the proposed $r-$refinement strategy. The flow is governed by the compressible, supersonic Euler equ., Fig. 1 shows an example pressure field and corresponding manufactured source term. The Onera M6 wing[a] is used as a more realistic and physically meaningful application example - Fig. 1c.



(a) *pressure (p)* (b) *energy src (f_e)* (c) *M6 wing Transonic flow and mesh.*

Figure 1: The 3D supersonic manufactured solution.

Acknowledgements

This work has been conducted within the About Flow project on Adjoint-based optimisation of industrial and unsteady flows, which has received funding from the European Unions Seventh Framework Programme for research, technological development and demonstration, Grant Agreement No. 317006.

References

[1] Pironneau, O., "On Optimum Design in Fluid Mechanics," Vol. 64, 1974, pp. 97–110.

[2] Fidkowski, K. J. and Darmofal, D. L., "Review of Output-Based Error Estimation and Mesh Adaptation in Computational Fluid Dynamics," Vol. 49, No. 4, 2011, pp. 673–694.

[3] Darmofal, D. and Venditti, D., "Anisotropic grid adaptation for functional outputs: application to two-dimensional viscous flows," *JCP*, Vol. 187, 2003, pp. 22–46.

[4] Ponsin, J., Fraysse, F., Gómez, M., and Cordero-Gracia, M., "An adjoint-truncation error based approach for goal-oriented mesh adaptation," *Aerospace Science and Technology*, Vol. 41, 2015, pp. 229–240.

[5] Dwight, R. P., "Robust mesh deformation using the linear elasticity equations," *Computational Fluid Dynamics 2006*, Springer, 2009, pp. 401–406.

[6] Tyson, W. C., Derlaga, J. M., Roy, C. J., and Choudhary, A., "Comparison of r-Adaptation Techniques for 2-D CFD Applications," *AIAA Paper*, Vol. 2611, 2015, pp. 22–26.

[7] Gugala, M. and Müller, J.-D., "Towards an output-based re-meshing for trubomachinery applications," *Proceedings ECCOMAS 2016 (in revision)*.

[8] Hascoët, L. and Pascual, V., "TAPENADE 2.1 user's guide," Technical Report 0300, INRIA, 2004.

[9] Roy, C. J., Smith, T. M., and Ober, C. C., "Verification of a compressible CFD code using the method of manufactured solutions," *energy*, Vol. 2, 2002, pp. s2.

[a] http://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing.html

Multi-disciplinary optimization of a compressor rotor subjected to ice impact using metamodelling

R. C. Schlaps*
Queen Mary University of London, UK

J. Ollar *
Altair Engineering, Leamington Spa, UK

S. Shahpar†
Rolls-Royce plc., Derby, UK

V. V. Toropov ‡
Queen Mary University of London, UK

Compressors of gas turbine engines are multi-disciplinary systems where the different disciplines are largely considered separately during the design process. In order to produce feasible designs, the components need to be re-designed several times which is time consuming. A multi-disciplinary design process enabling an integrated approach on each discipline is described in this paper. The main disciplines considered are aerodynamics and ice impact-worthiness for the front stage intermediate pressure compressor (IPC) rotor of a modern three-spool jet-engine.

Due to long lead times, ice impact analyses are not carried out until the aero design is reasonable mature. That allows usually only small changes to the rotor in order to achieve the impact requirements. Hence, the introduction of ice impact analysis in the earlier design stages provides higher flexibility for the designers and leads to better compressor performance. A fast thick-shell approach has been adopted to model the transient dynamics of the compressor rotor impacted by crystalline ice slabs from upstream stators. The aerodynamic performance is evaluated with 3D computational fluid mechanics (CFD).

The disciplines are linked using surrogate models which are built up with design of experiments (DoE) and response surface methods (RSM). The surrogate model allows identifying quickly suitable regions in the design space which is leading to improved rotor designs.

I. Introduction

Aircrafts are flying through cold and moist conditions during take-off and landing where the liquid water droplets are super-cooled which means that the water temperature is below -15C. The liquid super-cooled water droplets are entering the core engine and are freezing instantaneously when hitting cold engine surfaces such as stators and casings. The ice accretes to thick crystalline ice shells and eventually shed travelling downstream and impacting the downstream compressor rotor. Due to the high rotor speeds, the ice impact could cause severe mechanical damage if the impact-worthiness is not considered during the design process. In addition, the ice ingestion can affect the compressor aerodynamics causing a shift of the working line and stall margin when the ice melts and evaporate.[1] After a thorough literature search, a lack of suitable methods for ice impact on aeroengine rotors which are appropriate for optimisation has been identified. A model for transient impact dynamics has been developed and used for multi-disciplinary design optimisation.

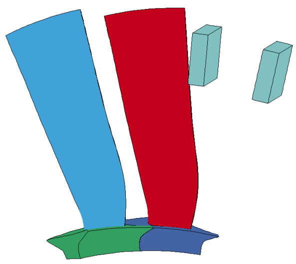


Figure 1. Ice impact model with two ice slabs and two blisk rotor blades.

*Marie Curie AMEDEO ITN Research Fellow.
†Associate Fellow, Team Lead 3D Geometry, Meshing and Optimisation.
‡Professor.Director of Centre for Simulation and Applied Mechanics , AIAA Senior Member.

A. Ice Impact model

A thick shell approach where the blade is modelled with a one layer brick mesh has been developed in order to reduce the computational time significantly without losing much accuracy. The explicit solver of LS-DYNA[2] is used to simulate the transient dynamics of the impact. In order to avoid highly distorted elements in the lagrangian model, element erosion is applied. In addition, the element erosion helps to model the slicing process more accurate. The impact model consist of two blades and two ice slabs (fig. 1). The size and position of the ice slabs has been chosen to represent a worst case scenario.

B. Aerodynamic model

The aerodynamic performance of the blade geometries is assessed via the Rolls-Royce in-house Reynolds-Averaged-Navier-Stokes solver Hydra.[3] It is an edge-based unstructured CFD code. The turbulence closure is done by using the Spalart-Allmaras turbulence model. The flow is considered to be fully turbulent in the CFD simulations.

For the mesh generation an automatic and robust mesh generation system is necessary avoiding any user interaction. Therefore, the Rolls-Royce mesh generator PADRAM[4] is enabling high quality block-structured meshes to be generated fully automatically.

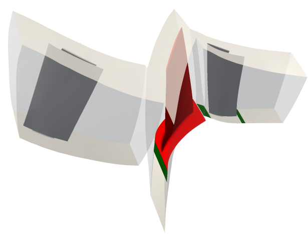


Figure 2. Rotor (red) embedded in a 1.5 stage setup with a shroud leakage model (green).

C. Parameterisation

The aerofoil section shapes are defined by the classical aerofoil parameterisation using the camberline angle Φ and thickness T distribution and the chord length c. Figure 3 shows the parameter definitions of a classical aerofoil parameterisation. The Rolls-Royce in-house tool Parablading is used to generate the aerofoils according to the prescribed parameters.

It can be imagined that the design space is large if all the available parameters for each section are used to define the aerofoil shape. In order to make the optimisation more efficient a reduction of the design parameters is necessary. An approach using B-splines to describe the design parameter distributions in radial and chordwise direction has been chosen. Overall 25 design parameters allowing to change the design with high flexibility has been used.

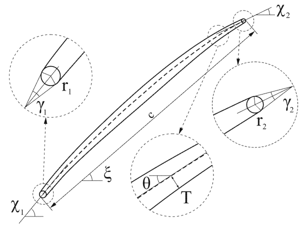


Figure 3. Classical aerofoil section parameterisation.

D. Optimisation method and strategy

Direct optimisation where every asked point from the optimiser is evaluated is rather expensive. Much effort has been made to develop optimisation strategies which require less amount of expensive simulations. In this study the mid-range approximation method (MAM)[5–11] has been used. The MAM is using a trust region approach where the optimisation problem is transformed to a sequence of sub-problems.

The optimisations were conducted using the Rolls-Royce SOPHY system.[12,13] The optimiser is communicating through Python scripts with the simulation codes in order to obtain the figure of merit. The Python scripts are able to execute multiple designs in parallel on a HPC cluster in order to speed up the optimisation.

II. Results

The results of the optimisation are shown in figure 4. The figure is showing the results normalised with the nominal values. The optimisation spent 8 iterations with 310 design evaluations in the aero discipline and 330 in the impact discipline. It was possible to increase the efficiency by 0.27% (fig. 4(a))) while reducing the damage due to ice impact. The figure of merit was reduced from an initially infeasible value of 1.203 to 1.003 which means that the constraint is still violated by negligible 0.3%. In addition, the constraint on the efficiency at the higher speed is also not fully satisfied in the final design being also 0.3% above the limit. All other constraints are satisfied which can be seen in figure 4(b).

Turbine Stator Well Geometry Benefits – Method Validation and Design Optimisation

Julien Pohl* and Harvey M. Thompson†
School of Mechanical Engineering, University of Leeds, Woodhouse Lane, LS2 7JT, Leeds, UK

Vincenzo Fico and Gary A. Clayton‡
Thermo-Fluids System, Rolls-Royce plc., Derby, UK

Engine components are commonly exposed to air temperatures exceeding the thermal material limit in order to increase the overall engine performance and to maximise the engine specific fuel consumption. To prevent the overheating of the materials and thus the reduction of the component life, an internal flow system must be designed to cool the critical engine parts and to protect them. As the coolant flow is bled from the compressor and not used for the combustion an important goal is to minimise the amount of coolant in order to optimise the overall engine performance.

During a typical flight cycle an aero-engine undergoes different operating conditions which cause varying temperatures, pressures, stresses and displacements to the engine components. From an engineering perspective, it is desirable to be able to accurately predict these behaviours in order to stay within the environmental and safety margins and to maximise component life. This also avoids costly experimental engine tests and increases the competitiveness of the aero-engine company in their market.

Predicting the metal temperatures is of paramount importance as they are a major factor in determining the component stresses and lives. In addition, as modern engines operate in ever harsher conditions due to efficiency requirements, the ability to predict thermal displacements becomes very relevant: on one hand, to prevent damage of components due to excessive rubbing, on the other hand, to understand how much air is flowing internally within the secondary air system for cooling and sealing purposes, not only in the design condition but throughout the engine life-span. In order to achieve this aero-engine manufacturers aim to use more and more accurate numerical techniques requiring multi-physics models, including thermo-mechanical finite elements and CFD models, which can be coupled in order to investigate small variations in temperatures and displacements.

This paper summarises the work carried out during the EU funded research project AMEDEO[1] and shows a practical application and extension of the methodology developed during the five year research programme MAGPI[2]. Extensive use is made of FEA (solids) and CFD (fluid) modelling techniques to understand the thermo-mechanical behaviour of a turbine stator well cavity, due to the interaction of cooling air supply with the main annulus. Previous work based on the same rig showed difficulties in matching predictions to thermocouple measurements near the rim seal gap[3-6]. In this investigation, further use has been made of existing measurements of hot running seal clearances in the rig. The structural deflections have been applied to the existing model to evaluate the impact in flow interactions and heat transfer[7,8].

In addition to a baseline test case without net ingestion, a case simulating engine deterioration with net ingestion is validated against the available test data, also taking into account cold and hot running seal clearances. Furthermore an additional geometry with a stationary deflector plate is modelled and validated for the same flow cases. Experiments as well as numerical simulations have shown that due to the deflector plate the cooling flow is fed more directly into the disc boundary layer, allowing more effective

*Communicating Author: j.pohl@leeds.ac.uk.
* Marie Curie Early Stage Researcher and PhD candidate.
† Supervising Professor in Computational Fluid Dynamics, School of Mechanical Engineering.
‡ Project Partners and Specialists in CFD and Thermal Analyses of Secondary Air Systems.

use of less cooling air, leading to improved engine efficiency. Therefore, the deflector plate geometry is embedded in a CFD-based automated optimisation loop to further reduce the amount of cooling air[9]. The optimisation strategy concentrates on a flexible design parameterisation of the cavity geometry with deflector plate and its implementation in an automatic 3D meshing system with respect of finally executing an automated design optimisation. Special consideration is given to the flexibility of the parameterisation method in order to reduce design variables to a minimum while also increasing the design space flexibility & generality.

The parameterised geometry is optimised using a metamodel-assisted approach based on regressing Kriging in order to identify the optimum position and orientation of the deflector plate inside the cavity. The outcome of the optimisation is validated using the benchmarked FEA/CFD coupling methodology.

Acknowledgements

The present investigations were supported by the European Commission within the AMEDEO EU initial training network and within the Research Project MAGPI, for the use of the acquired test data. A special mention must be made to Rolls-Royce staff Christopher Barnes, Shahrokh Shahpar and Jeff Dixon for their support and advice.

References

[1]AMEDEO FP7 Marie Curie Initial Training Network, 2013. “Aerospace Multidisciplinary Enabling Design Optimisation”. Grant reference: 316394.

[2]MAGPI, 2006. “Main Annulus Gas Path Interaction - Specific Targeted Research Project”. Proposal Contract no.: 30874.

[3]Dixon, J. A., Guijarro-Valencia, A., Coren, D., D., and Long, C., 2014. “Main Annulus Gas Path Interactions - Turbine Stator Well Heat Transfer”. Journal of Turbomachinery, 136, February, pp. 021010–1–16

[4]Smith, P. E. J., Mugglestone, J., Tham, K. M., Coren, D. D., and Long, C. A. “Conjugate Heat Transfer CFD Analysis in Turbine Disc Cavities”. In ASME Turbo Expo 2012, GT2012-69597.

[5]Andreini, A., DaSoghe, R., and Facchini, B., 2011. “Turbine Stator Well CFD Studies: Effects of Coolant Supply Geometry on Cavity Sealing Performance”. Journal of Turbomachinery, 133, April, pp. 021008–1–11.

[6]Lück, H., Schäfer, M., and Schiffer, H.-P., 2014. “Simulation of Thermal Fluid-Structure Interactions in Blade-Disc Configuration of an Aircraft Turbine Model”. In ASME Turbo Expo 2014, GT2014-26316.

[7]Pohl, J., Fico, V. and Dixon, J. A., 2015. “Turbine Stator Well Cooling – Improved Geometry Benefits”. In ASME Turbo Expo 2015, GT2015-42658.

[8]Pohl, J., Thompson, H. M., Guijarro Valencia, A., Lopez Juste, G., Fico, V. and Clayton, G. A., 2016. “Structural Deflection’s Impact in Turbine Stator Well Heat Transfer”. In ASME Turbo Expo 2016, GT2016-56428

[9]Pohl, J., Thompson, H. M., Schlaps, R. C., Shahpar, S., Fico, V. and Clayton, G. A., 2016. "Innovative Turbine Stator Well Design Using Design Optimisation". In 16th ISROMAC & 1st ISIMET.

Use of layout optimization to identify optimal bracing systems in buildings

Many studies of the efficiency of lateral stability systems for buildings have been undertaken over the years. However, current design procedures appear not to be underpinned by clear fundamental principles. In this study, arrangements of bracing members are sought using the well-known layout optimization technique. The efficacy of the techniques is demonstrated by using it to identify the optimal arrangement of bracing and column elements in a small-scale building. When member buckling effects were ignored, the stiffness to material usage ratio of a braced bay was found to be 32% greater than when traditional diagonal cross-bracing was used – or 19% greater when buckling effects were accounted for.

Hongjia Lu[*], Matthew Gilbert and Andrew Tyas
Department of Civil and Structural Engineering, University of Sheffield, S1 3JD, UK

I. Introduction

The bracing system is an indispensable part of any building. This is particularly important in a tall building structure where the requirements for lateral stiffness may govern the layout of the whole building. Consequently, the design of the layout of the bracing required to resist lateral wind loading is an active area of structural engineering research. Studies of the optimality of bracing systems has typically involved the use of a given, fixed, topology, and then used either geometry or size optimization to identify the best solution. For example, Moon (2007) analysed diagrid bracing systems, using the angles between bracing members as parameters which were varied to optimize the lateral stiffness of the structure. Baker (1990) presented a bracing size optimization technique based on energy methods. Others have used continuum optimization techniques (Stromberg 2012; Liang 2000) or evolutionary optimization approaches (Richardson 2013). What has rarely been considered is the use of efficient, classical discrete layout optimization methods to identify benchmarks against which the performance of different design solutions can be judged.

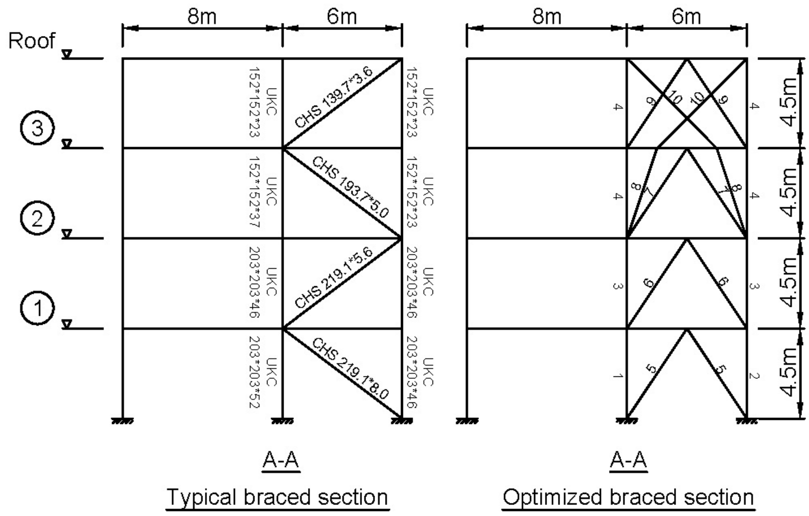
The mathematical basis for the problem of finding the structural layout consuming least material was developed by Michell (1904). Whilst the analytical approach put forward by Michell provides a strong foundation for the structural optimization field, it is rarely helpful in a specific structural design application, because of the difficulty in finding an optimum structure for a given set of loading conditions. However, Michell's approach readily lends itself to numerical implementation, where the optimal layouts of discrete bar members are found from a “ground structure” comprising all possible interconnections between discrete node points within a design space. Using linear programming (LP) techniques numerical results can be found which provide close approximations of ideal Michell forms for a given problem (Dorn 1964; Hemp 1973). However, despite the efficiency of LP, even this method become computationally intractable for large-scale problems, due the number of potential members in the ground structure. Consequently, an improved, iterative “member adding” method was developed by two of the present authors, which dramatically decreases the computational cost, allowing very large scale numerical problems to be addressed (Gilbert and Tyas 2003). In this study, this approach is used to investigate optimal bracing layouts for different scenarios.

Previous workers have often considered the optimization of bracing systems where a beam and column framing system has already been designed to carry gravity loads, and where the load cases considered for bracing design involve only lateral loads. Stromberg et al. (2012) considered the effect of the stiffness of the columns on the optimal bracing layout, considering a single structural bay aspect ratio in their study. Liang et al. (2000) considered several bay aspect ratios, and found that the column and beam sizes determined from the gravity load analysis were adequate for the lateral load cases and geometries investigated. This raises two questions: (i) if the columns and beams designed for gravity loads can be assumed to have sufficient reserves of strength to act as part of the lateral stability system, can we find general rules governing the layout of the latter?; (ii) what are the optimal bracing layouts for frames where the structural layout is optimized for both gravity and lateral load cases, and combinations thereof. Both these questions are addressed in the present study, though here the focus is on (ii).

[*]PhD student, Email: hlu15@sheffield.ac.uk

II. Optimal bracing layout for small-scale building

A design example involving a typical naturally ventilated office considered by Brettle and Brown (2009) is here used to demonstrate the potential for layout optimization to be applied to building framing system design. The layouts of columns and bracing members are first optimized using plastic methods, using British Standard load cases (i.e. both vertical and lateral loads are considered). Bracing and column members are then resized if necessary to ensure buckling failure does not occur. The section details of both a traditionally designed and optimized braced bay are shown in Figure 1. It is worth noting that, during the optimization process, beams are treated as pre-existing members with infinite strength since they are always designed only according to gravity loading.



1	UKC 203*203*52
2	UKC 203*203*46
3	UKC 152*152*37
4	UKC 152*152*23
5	CHS 168.3*8.0
6	CHS 168.3*5.0
7	CHS 139.7*5.0
8	CHS 76.1*2.9
9	CHS 48.3*2.6
10	CHS 60.3*2.6

Section data of layout optimized braced bay

Figure 1 Optimization of a braced bay in a small-scale multi-storey building

The two braced bay can be compared in terms of both total material usage and lateral stiffness. It was observed that the stiffness to material cost ratio was 32% greater when layout optimization was employed, initially ignoring buckling considerations. When buckling considerations were included the volume of material required increased and the stiffness to material cost ratio was 19% greater when layout optimization was employed. Second order effects can be checked using the parameter α_{cr} defined in Eurocode 3. With the sections shown in Figure 1, each α_{cr} of floors in the optimized braced bay is larger than the threshold value defined by the Eurocode (i.e. 10), which means second order effects can be neglected.

III. Conclusions

Layout optimization appears to provide a useful means of identifying optimal bracing systems in buildings. Future work will focus on extending the layout optimization technique so that rigid-jointed framing elements can also be modelled.

IV. References

Baker (1990), *In: 4th world congress on tall buildings, pp 868-875.*
Brettle and Brown (2009), *The Steel Construction Institute.*
Dorn WS, Gomery RE & Greenberg HJ (1964), *J. de Mecanique* 3.
Gilbert M & Tyas A (2003), *Engineering Computations*, 20.
Hemp, WS (1973), *Optimum Structures*, Clarendon Press.
Liang (2000), *Journal of Structural Engineering*, 126.
Michell (1904), *Philosophical Magazine*, 8.
Moon (2007), *Structural Design of Tall and Special Buildings*, 16.
Richardson (2013), *Automation in construction*, 32.
Stromberg (2012), *Engineering Structures*, 37.

SOD: A Tool Auto-Generates the Preliminary Structural Design of Steel-Composite Office Buildings

Li Huang* and Martin Mensinger†
Chair of Metal Structures, Technische Universität München, Munich, 80333, Germany

At conceptual design phase of a building project design space exploration is often used to seek better design solutions. The number of design variants to be compared is limited by the analyzing time of each variant. Comparison of the engineering performance and the material consumption is usually based on preliminary structural designs or pure engineering experience, which we think can be improved. A practical assistant tool Sustainable Office Designer (SOD) is presented in this paper. It can fast auto-generate preliminary structural designs of steel-composite office buildings using an optimization approach.

Nomenclature

GA	=	Genetic Algorithm
SOD	=	Sustainable Office Designer
VAD	=	Volumetric Architectural Design
PSD	=	Preliminary Structural Design
PSM	=	Parametric Structural Model

I. Introduction

THE conceptual design phase is most crucial in the sense of achieving an optimal building design, because that the most significant decisions are made in this phase, such as the number of floors, the overall shape, the construction type (concrete, steel or wood), etc. These decisions restrict the possible improvement of the final design. Better decisions can be obtained by design space exploration, i.e. by comparing different design variants. Because engineering structural design is a complex task, at conceptual design phase the design models are usually limited to volumetric architectural design (VAD). The comparison of material consumption, which can be used to calculate the life-cycle assessment indicators and construction cost, are often based on preliminary structural designs (PSDs) or pure engineering experience. This limits the number of possible design variants and construction types to be considered in this early stage.

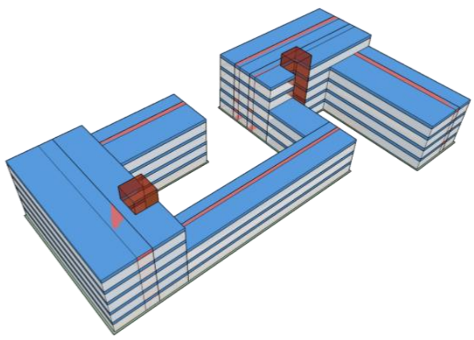


Figure 1: Volumetric Architectural Design

Therefore, shorten the creation of the PSD for each variant can enlarge the number of variants for comparison and lead to better decisions. Optimization approach is employed to seek optimal solution of engineering design problems for decades. Structural design is one of the complex engineering design problem. Former researches[1,2,3,4] shows great successes using genetic algorithms (GAs) to find optimized structural design solutions for concrete and steel constructions due to its robustness, flexibility and simplicity in implementation.

In this paper the principle and result of SOD is presented. It is an application to auto-generate PSD for low-rise steel-composite office buildings[5]. It employs a fast structural optimization approach. The presented study focuses on buildings having a floor-layout comprising one or more rectangular shapes, since rectangle is the most common shape in office buildings.

Communicating Author: li.huang@tum.de
* Research Assistant at the Chair of Metal Structures, Technische Universität München
† Head of the Chair of Metal Structures, Technische Universität München

II. Volumetric Architectural Design and Preliminary Structural Design

A. Volumetric Architectural Design

The volumetric model (see Fig. 1) is used in conceptual architectural design for visualization and to perform shading and building energy and other analyses.

Necessary information for generating PSD can be obtained from or defined in VAD, such as:

the shape of the floor layout,
the number of stories for each building part,
the story room height and the additional story height that allows the placement of structural elements (such as beams and slabs) and engineering components (such as ventilation and celling),
the façade width that restrict the column spacing and location of intermediate columns.

Figure 2: Preliminary Structural Design

B. Auto-generated Preliminary Structural Design

From the VAD we create the Parametric Structural Model (PSM) and then use a GA to find the optimized structural solution (see Fig. 2). To perform the optimization, we need to calculate the objective value and the penalty value. The objective value is calculated as weighted summation of the material consumption, which is easily calculated from the structural model. The penalty value is used to verify the structural design and eliminate the infeasible structural solutions. A simplified structural analysis routine is implemented to verify the structural components including continuous composite primary beams, composite secondary beams and steel columns w.r.t. "Eurocode 3" and "Eurocode 4". Composite slabs are verified against pre-calculated design tables.

SOD only performs pre-structural analysis in order to gain computation speed. Therefore, in the calculations only vertical loads are taken into account. Horizontal loads are not considered. Due to the large number of possible vertical bracing options and the fact that additional masses for bracing and additional supporting elements are often independent of the structural design choices, the calculation of vertical bracing is neglected.

III. Result

For a building with floor height comprising six rectangular parts: 12m × 8m × 3 stories, 36m × 18m × 5 stories, 24m × 12m × 4 stories, 36m × 17m × 5 stories, 48m × 8m × 3 stories and 30m × 15m × 4 stories using the composite slab Holorib 120mm/140mm/160mm/180mm, Cofraplus 120mm/140mm/160mm/180mm or the TOPFloor system, SOD can generate an optimized preliminary structural design within several minutes on a normal laptop powered by Intel CPU i7 4900MQ and 16GB memory.

IV. Conclusions

SOD, an application of a fast optimization approach, has been implemented in this work to auto-generate preliminary structural design of steel-composite office buildings in early conceptual design phase. It uses a generic algorithm to generate optimized structural design based on the rectangular PSM. Simplified structural analysis is performed to verify the structural components. It has been shown that the PSD can be automatically generated in very short time to enlarge the number of deign variants for comparison. The further work is to extend the approach for more complex shapes besides rectangles and to take into account building cores.

References

[1]Rafiq, M.Y., Mathews, J., Bullock, G.N., "Conceptual Building Design – An Evolutionary approach" ASCE J. Comput. Civil Eng., 17(3), pp. 150-158, 2003.

[2]Turrin, M., von Buelow, P., Stouffs, R., "Design explorations of performance driven geometry in architectural design using parametric modeling and genetic algorithms" J. Adv. Eng. Inf., 25, pp. 656-675, 2011.

[3]Shaw, D. J., "Geometric representations for conceptual design using evolutionary algorithms" PhD Thesis, Cardiff University, 2006

[4]Nimtawat, A., and Nanakorn, P., "A genetic algorithm for beam-slab layout design of rectilinear floors" Eng. Struct. 32(11), pp. 3488-3500, 2010

[5]Mensinger, M., Huang, L., Hogger, H., "Sustainable Design and optimization of office buildings in steel and composite construction" Concepts and Methods for steel intensive building projects, H. Koukkari, L.Braganca, S. Boudjabeur (Editor.), ECCS TC14 Report N°130, 2012.

Evaluating sensitivity of stochastic search techniques to profile list ordering in discrete sizing of steel frames

Saeid Kazemzadeh Azad

Assistant Professor, Atilim University, Department of Civil Engineering, Ankara, Turkey

Abstract

In practical design of steel frame structures mostly the structural members are selected from a list of available sections resulting in a discrete optimization problem. In recent decades, considerable research work has been devoted to development of efficient algorithms for tackling discrete sizing optimization problems of steel structures. In spite of the large number of developed algorithms, only a few studies address non-algorithmic issues, such as profile list ordering, affecting the general performance of the algorithms in the course of optimization. This study is an attempt to investigate the effect of the order of sections in a profile list on general performance of stochastic search algorithms or the so called metaheuristics. To this end, sensitivity of recently proposed metaheuristic algorithms to different profile lists is investigated through practical sizing optimization of real size steel frame structures according to AISC-LRFD specification. The numerical results reveal the effect of profile list ordering on optimality of final solutions as well as convergence rate of the investigated algorithms.

Key words: Discrete optimization; steel frames; metaheuristic algorithms; adaptive dimensional search; big bang-big crunch algorithm; AISC-LRFD.

Evolutionary Crashworthiness Topology Optimization of Thin-Walled Structures

Mariusz Bujny*
Technische Universität München, Munich, 80333, Germany

Nikola Aulig†
Honda Research Institute Europe GmbH, Offenbach/Main, 63073, Germany

Markus Olhofer‡
Honda Research Institute Europe GmbH, Offenbach/Main, 63073, Germany

and

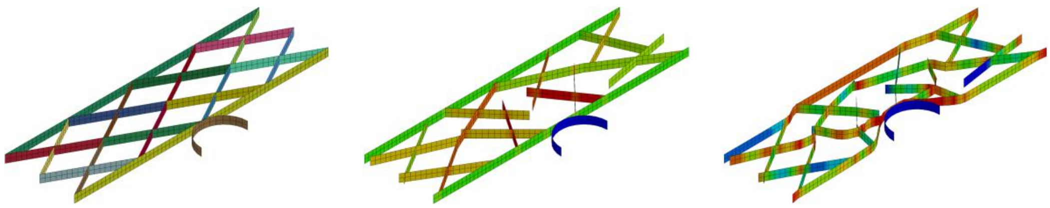
Fabian Duddeck§
Technische Universität München, Munich, 80333, Germany

As in many other disciplines, also in crashworthiness, the extensive growth of computers' power led to the development of techniques for numerical simulations. In particular, this allows to use numerical optimization methods to develop better structures and shorten the vehicle design cycle, what is a must in case of the hard competition on the car market. One of the most important tasks realized by optimization algorithms is the identification of optimal structural concepts in early phases of the design process. This problem is addressed by topology optimization techniques[1], which aim to develop optimal structural concepts within a defined design space and under specified boundary conditions. Nowadays, those methods play a vital role in many branches of industry in fast prototyping of efficient mechanical structures. Nevertheless, due to the complexity of crash phenomena and considerable simplifications made in most of the optimization approaches[2-9], the use of crashworthiness topology optimization techniques is still limited and new methods have to be developed.

A basis for most of the state-of-the-art methods for crashworthiness topology optimization, form so-called voxel elements, being three-dimensional, regular brick finite elements. The basic idea in such cases is stated as follows: Firstly, define the design domain in the space that is not occupied with non-structural vehicle elements such as wheels, engine, etc.; Secondly, fill the volume of the design space with voxels; Thirdly, eliminate redundant voxels by an iterative numerical optimization procedure. This results in creation of so-called zigzag structures that can be used as a reference for positioning of the structural beams. Such a design is assumed to be optimized with respect to the given objective (e.g. energy absorption, plastic deformation, etc.), although due to the heuristic assumptions this is only true for selected use cases. Additionally, important vehicle body components are made of thin-walled sheet metal structures. In such a case plastic buckling is the principal phenomenon that influences the crashworthiness behavior. In the optimization process based on voxel elements, structures made of thin metal sheets cannot be obtained, which leads to completely different phenomena, not corresponding to the buckling of thin-walled structures. As a result, the use of an optimized design obtained from any voxel-based optimization method as an inspiration for final thin-walled structure is questionable and alternative methods have to be developed.

Communicating Author: mariusz.bujny@tum.de.
* Doctoral Candidate, Chair of Computational Mechanics.
† Scientist, Complex System Optimization & Analysis.
‡ Chief Scientist, Complex System Optimization & Analysis.
§ Professor for Computational Mechanics, Chair of Computational Mechanics.

We propose a novel approach using evolutionary algorithms for optimization of thin-walled structures. Unlike in the other approaches for crashworthiness topology optimization of thin-walled structures[9-11], in evolutionary optimization methods, no heuristic assumptions about the properties of the optimization problem are made, and therefore, any actual quantifiable user defined objective function can be optimized directly. For evaluation of the method, a 2D transverse bending of a rib-reinforced thin-walled structure is considered. Parameterization of the design is realized through defining the position, orientation, length and thickness of each reinforcing rib. The ribs can cross each other and join if they are sufficiently close to each other. As an optimization method both standard Evolution Strategy (ES) and the state-of-the-art Covariance Matrix Adaptation Evolution Strategy (CMA-ES) are used and their performance is compared. The results show that evolutionary optimization algorithms can be efficiently used for crashworthiness topology optimization of thin-walled structures.



a). Initial design. **b). Thickness distribution for the optimized design.** **c). Effective stress at the end of the crash simulation.**

Figure 1. Initial layout of the reinforcing ribs and the best design obtained with the Covariance Matrix Adaptation Evolution Strategy (CMA-ES).

References

[1]Bendsøe, M. P., and Sigmund, O., "Topology Optimization", Springer Berlin Heidelberg, Germany, 2004.

[2]Christensen, J., Bastien, C., and Blundell, M. V., "Effects of roof crush loading scenario upon body in white using topology optimization", *International Journal of Crashworthiness*, 17(1):29–38, 2012.

[3]Duddeck, F., and Volz, K., "A new topology optimization approach for crashworthiness of passenger vehicles based on physically defined equivalent static loads", *Proceedings of the ICRASH conference*, Milano, Italy, 2012.

[4]Park, G.-J., "Technical overview of the equivalent static loads method for non-linear static response structural optimization", *Structural and Multidisciplinary Optimization*, 43(3):319–337, 2010.

[5]Fredricson, H., Johansen, T., Klarbring, A., and Petersson, J., "Topology optimization of frame structures with flexible joints", *Structural and Multidisciplinary Optimization*, 25(3):199–214, 2003.

[6]Pedersen, C. B. W., "Topology optimization design of crushed 2d-frames for desired energy absorption history", *Structural and Multidisciplinary Optimization*, 25(5-6):368–382, 2003.

[7]Mozumder, C. K., "Topometry optimization of sheet metal structures for crashworthiness design using hybrid cellular automata", PhD thesis, University of Notre Dame, USA, 2010.

[8]Patel, N. M., "Crashworthiness Design Using Topology Optimization", PhD thesis, University of Notre Dame, USA, 2007.

[9]Ortmann, C., and Schumacher, A., "Graph and heuristic based topology optimization of crash loaded structures", *Structural and Multidisciplinary Optimization*, 47(6):839–854, 2013.

[10]Hunkeler, S., "Topology Optimisation in Crashworthiness Design via Hybrid Cellular Automata for Thin-walled Structures", PhD thesis, Queen Mary University of London, UK, 2013.

[11]Duddeck, F., Hunkeler, S., Lozano, P., Wehrle, E., and Zeng, D., "Topology Optimization for Crashworthiness of Thin-walled Structures under Axial Impact Using Hybrid Cellular Automata", *Structural and Multidisciplinary Optimization*, 2015.

Automatic adjoint formulation with customized objective functions in an industrial CFD framework

Yingchen Li [*] , Kilian Claramunt [†] and Charles Hirsch [‡]
Numeca Int., Brussels, 1170, Belgium

The formulation of continuous adjoint method depends on the objective function. The present paper develops a flexible new way to customize the objective function within the frame of the industrial CFD code FINE/Open with OpenLabs™. The formulation of the adjoint equations, boundary conditions and sensitivity gradients is automatically derived by the system without user intervention. Several examples are presented to show the capability of interpreting various objective functions in a variety of applications.

I. Abstract

ADJOINT methods receive growing attention in aerodynamic shape optimization and mesh adaptation, due to its reduced computational cost in obtaining sensitivity gradient in a N-dimensional parameter space. However, the highly intrusive adjoint formulation depends on the objective function, making most of the adjoint solvers either limited to a list of predefined objective functions or requiring a dedicated re-programming to adapt to a new objective function.

The present paper develops a flexible new way to customize the objective function within the frame of the industrial CFD code FINE/Open with OpenLabs™. The engineer can define his objective functions by a text file, and the adjoint system (equations, boundary conditions as well as sensitivity gradients) will be adapted automatically by OpenLabs. OpenLabs is a user friendly component of FINE/Open, allowing the user to develop its own physical models, boundary or initial conditions, add new transport equations, …, without explicit programming.

The methodology is based on the continuous adjoint method. The general formulation of the adjoint system is firstly derived for the given prototype of the objective function, which involves both surface and volume integrals of flow quantities. Then, symbolic manipulations on the new objective functions are performed by OpenLabs, including integration by parts, Leibniz rule, differentiation using chain rule and algebraic manipulations. Both analytical and numerical differentiations, based on complex variable methods, are applied. Through the symbolic manipulations, the contributions of the objective functions to the adjoint system are formulated by OpenLabs, including 1) adjoint boundary condition; 2) source terms of adjoint equation (if any); 3) sensitivity gradients with respect to boundary conditions and to the surface mesh nodes. A shared library is then automatically created and plugged into the adjoint solver of FINE/Open with OpenLabs™. All steps above are performed fully automatically, without user intervention.

The objective functions can be defined as integrals over boundaries or domains, whose integrand can be defined as the algebraic combination of the flow variables and their 1st and 2nd-order spatial derivatives. In addition, the objective function can also be defined as linear, product or ratio combination of the integrals.

Several examples are presented in the paper to show the capability of FINE/Open with OpenLabs™ of interpreting various objective functions:

1. Validation: pressure inverse design.
2. Single objective: minimize entropy.
3. Multi objective: minimize mass flow averaged entropy with exit flow angle constraints.
4. Multi objective: minimize total pressure losses and minimize exit whirl angle.
5. Multi objective: maximize average heat transfer coefficient and minimize mass flow averaged total pressure losses.

Communicating Author: charles.hirsch@numeca.be.
[*] Responsible for OpenLabs, Multiphysics CFD Group, Chaussée de la Hulpe, 187-189, 1170 Brussels, Belgium
[†] Head of Multiphysics CFD Group, Chaussée de la Hulpe, 187-189, 1170 Brussels, Belgium
[‡] Professor, Founder and President, Chaussée de la Hulpe, 187-189, 1170 Brussels, Belgium

Acknowledgments

The work described in this paper has been made possible by the fruitful and stimulating collaboration with the Laboratory of Prof. Kyriacos Giannakoglou and his team at the National Technical University of Athens (NTUA).

References

[1]M. B. Giles, N. A. Pierce, "Adjoint equations in CFD: Duality, boundary conditions and solution behavior," AIAA-97-1850, 1997.

[2]D.I. Papadimitriou, K.C. Giannakoglou, "A Continuous Adjoint Method with Objective Function Derivatives Based on Boundary Integrals for Inviscid and Viscous Flows," Computers & Fluids, Vol. 36, 2007, pp., 325, 341.

[3]J.R.R.A. Martins, P. Sturdza and J.J. Alonso, "The Connection between the Complex-step Derivative Approximation and Algorithmic Differentiation," AIAA-2001-0921, 2001.

[4]FINE™/Open with OpenLabs™ manual (http://www.numeca.be)

Adjoint Solver for Commercial Computational Fluid Dynamics Codes using Algorithmic Differentiation Tools

Zahrasadat Dastouri [*] and Uwe Naumann [†]

Software and Tools for Computational Engineering (STCE), RWTH Aachen University, 52074 Aachen, Germany

The main topic of this paper is the application of discrete adjoint method in large legacy Computational Fluid Dynamics (CFD) codes using Algorithmic Differentiation tools (AD). It provides an adjoint solver for generating accurate and detailed sensitivities for shape optimization problems in industrial CFD applications.

I. Introduction

Algorithmic Differentiation (AD)[1,2] employs the rules of differential calculus in an algorithmic manner to determine accurate derivatives of a function defined by computer programs. For a given implementation of the flow model $F : \mathbb{R}^n \to \mathbb{R}^m$ over a computational grid, the computer program is developed to simulate the functional dependence of one or more objectives $\mathbf{y} \in \mathbb{R}^m$ on a potentially large number of input variables $\mathbf{x} \in \mathbb{R}^n$ by simulation of, $F : \mathbb{R}^n \to \mathbb{R}^m, \mathbf{y} = F(\mathbf{x})$. AD enables us to compute the corresponding derivatives $\frac{\partial \mathbf{y}}{\partial \mathbf{x}}$ in forward (forward mode) or backward (reverse mode). For a given implementation of the primal function F, the function $F_{(1)} : \mathbb{R}^n \times \mathbb{R}^m \to \mathbb{R}^n \times \mathbb{R}^m$, defined as

$$(\mathbf{y}, \mathbf{x}_{(1)}) = F_{(1)}(\mathbf{x}, \mathbf{y}_{(1)}) \equiv \left(F(\mathbf{x}), \left(\frac{d\mathbf{y}}{d\mathbf{x}} \right)^T \cdot \mathbf{y}_{(1)} \right) \tag{1}$$

is referred to as the *adjoint* model of F. The adjoint model implementation yields the objective $\mathbf{y}$ and the product $\mathbf{x}_{(1)} \equiv \nabla F(\mathbf{x})^T \cdot \mathbf{y}_{(1)}$ of its transposed jacobian at the current point $\mathbf{x} \in \mathbb{R}^n$.

There are two main methods for implementing AD: by source code transformation (S-T) or by use of derived data types and operator overloading (O-O). In O-O AD the code segments and arguments of the primal code are stored inside a memory structure called tape during the forward run of the primal. In reverse mode the stored values on the tape are interpreted to get the resulting adjoints, while the S-T approach parses the code at compile time and generates the actual derivative code.

Prior to this paper[3],[4] we have discussed the implementation of the AD tool `dco/fortran`[a] *(Derivative Code by Overloading in Fortran)* to an unstructured pressure-based steady Navier-Stokes solver. The solver is an incompressible flow solver with cell-centered storage and face-based residual assembly[7] uses the SIMPLE[6] pressure correction algorithm. We have addressed proper solution algorithms adapted to the code for the improvement of efficiency of the adjoint code by optimizing the checkpointing scheme for the iterative solver and development of symbolically differentiated of linear solver. In addition, we investigated the benefit of the reverse accumulation technique[8] for the fixed point iterative construction in the primal code.

Moreover, in [5] we have discussed an hybrid approach by combining the flexibility and robustness of operator overloading with the efficiency of source transformation by coupling `dco/fortran` and `TAPENADE`.[9] The latter was used for the derivation of computationally expensive kernels. Our emphasis was to automate the implementation of an adjoint software to decrease the development time of the differentiated code. Computational efficiency is proved through demonstration examples.

In this paper, based on the experiences on applying AD on a medium size solver as a benchmark, reverse mode of AD has been applied in the context of shape optimisation for fluid dynamics systems to a complex commercially licensed CFD code, CFD-ACE+.[10] The multi-physics ACE+ solver has many features for flow modeling including laminar and turbulence flow under steady transient states. The functionalities enable decouple or coupled pressure based flow solvers simulations.

---

[*] PhD Candidate, Software and Tools for Computational Engineering (STCE), RWTH Aachen University, 52074 Aachen, Germany.

[†] Professor, Software and Tools for Computational Engineering (STCE), RWTH Aachen University, 52074 Aachen, Germany.

[a] developed at the institute a*Software and Tools for Computational Engineering* at RWTH Aachen University implementing AD by overloading in Fortran[1]

II. Development a the Discrete Adjoint Model for Commercial CFD CODE CFD-ACE+

Required elements for adjoint solver are utilized in CFD solution to get discrete adjoint version of the CFD code using Operator Overloading (O-O) and Source Transformation (S-T) AD tools. In CFD application, the resulting memory usage of the black-box adjoint approach is not acceptable for real world problems. Therefore, the need for efficient solution techniques arising from the PDE discretization of Navier-Stokes system has been emphasised and discussed for the problems considered such techniques. This includes symbolically differentiating the linear solver in the adjoint code and employing optimized checkpointing scheme and reverse accumulation for fixed point iteration (SIMPLE) loop. A robust and efficient adjoint solver is constructed applying O-O or S-T approaches and the combination of these tools for computationally expensive part of the solver with the objective of minimizing the development time to get the adjoint code. In both cases the utility of the approaches has been demonstrated by numerical experiments and relevant flow test cases to reflects and compare the performance assessment of the adjoint solver. A significant contribution of this work is to investigate the properties that the adjoint systems generated by AD tools posses in a complex legacy commercial solver compared to the original CFD solution. This leads to the conclusion on how to deal with the complexity that the developer must carry out to construct an effective CFD adjoint solver that performs acceptable for industrial problems and applications.

References

[1] U. Naumann. The Art of Differentiating Computer Programs. An Introduction to Algorith- mic Differentiation. SIAM, Philadelphia, 2012.

[2] A. Griewank and A. Walther. Evaluating Derivatives: Principles and Techniques of Algo- rithmic Differentiation. SIAM, Philadelphia, Jan 2000.

[3] Z. Dastouri, J. Lotz, and U. Naumann. Development of a discrete adjoint cfd code using al- gorithmic differentiation by operator overloading. In OPT-i: An International Conference on Engineering and Applied Sciences Optimization, Athens, 2014. National Technical University of Athens.

[4] Z. Dastouri and U. Naumann. Improving efficiency of a discrete adjoint cfd code for design optimization problems. In EUROGEN: Proceeding of International Conference on Evolutionary and Deterministic Methods for Design, Optimization and Control with Applications to Industrial and Societal Problems, Glasgow, UK, 2015.

[5] Z. Dastouri, S.M. Gezgin and U. Naumann. A Mixed Operator Overloading and source transformation approach for adjoint CFD computation. In Eccomass: Proceeding of European Congress on Computational Methods in Applied Sciences and Engineering, Crete Island, Greece, 2016.

[6] S.V. Patankar and D.B. Spalding. A calculation procedure for heat, mass and momen- tum transfer in three-dimensional parabolic flows. International Journal for Heat Mass Transfer, 15(10):17871806, 1972.

[7] D. Jones, F. Christakopoulos, and J-D. Muller. Preparation and assembly of adjoint cfd codes. Computers and Fluids, 46(1):282286, July 2011.

[8] B. Christianson. Reverse accumulation and attractive fixed points. Optimization Methods and Software, 3:311326, 1994.

[9] L. Hasco et and V. Pascual. The Tapenade Automatic Differentiation tool: Principles, Model, and Specification. ACM Transactions On Mathematical Software, 39(3), 2013.

[10] CFD-ACE+ Advanced CFD and Multiphysics Solver, ESI Group, http://www.esi-cfd.com/

Experiments on Checkpointing Adjoint MPI Programs

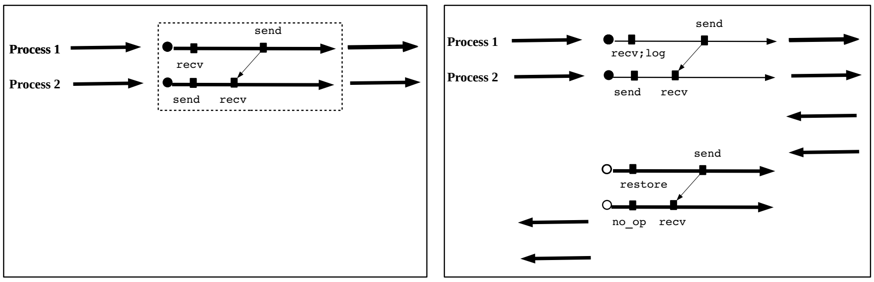
A.Taftaf

INRIA, Sophia-Antipolis, France, Elaa.Teftef@inria.fr

Checkpointing is a classical strategy to reduce the peak memory consumption of the adjoint. Checkpointing is vital for long run-time codes, which is the case of most MPI parallel applications. However, the presence of MPI communications seriously restricts application of checkpointing. In the most popular approach, a number of restrictions apply on the form of communications that occur in the checkpointed piece of code. In previous work, we proposed a general technique that lifts this restriction. This technique (called "receive-logging") is based on logging the values received, so that no duplicated communication is needed. However, the message logging makes it more costly than the popular approach. We proposed a refinement to our technique to duplicate communications whenever it is possible, so that the refined receive-logging now encompasses the popular approach. In this work we see checkpointing MPI parallel programs from a practical point of view. We discuss an important question about the choice of the checkpointed pieces. We validate our theorical results on a representative code in which we perform various choices of checkpointed pieces. We apply the refined receive-logging to these checkpointed pieces and we quantify the expenses in terms of memory and computation time for each resulting checkpointed adjoint.

Checkpointing is a classical technique to mitigate the overhead of adjoint Algorithmic Differentiation (AD). In the context of source transformation AD with the Store-All approach, checkpointing[1] reduces the peak memory consumption of the adjoint, at the cost of duplicate runs of selected pieces of the code. Checkpointing is vital for long run-time codes, which is the case for most MPI parallel applications. However, the presence of MPI communications seriously restricts application of checkpointing.
In most attempts to apply checkpointing to adjoint MPI codes (the "popular" approach), a number of restrictions apply on the form of communications that occur in the checkpointed piece of code. In particular, both ends of each communication must belong to the same checkpointed piece and the non blocking routines and their waits must be checkpointed together. If only one end is contained in the checkpointed piece of code, the resulting adjoint fails.
In previous work[2], we proposed a technique to apply checkpointing to adjoint MPI codes. This technique (called "refined receive-logging") is more general than the popular approach, i.e. it imposes less restrictions. The main idea of this technique (see figure 1) is to duplicate every communication whose ends belong to the checkpointed piece and to apply "receive-logging" to all the remaining communications, which are actually ends of communications whose other ends are outside the checkpointed piece (we call them "orphan communications"). Applying the receive-logging to one end of a communication means that:

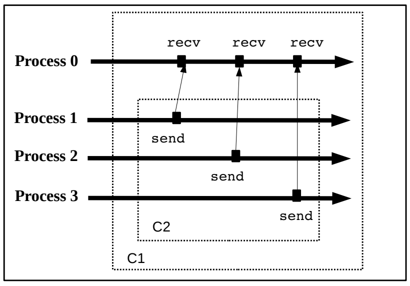
- During the first execution of the checkpointed piece, this end of communication is executed normally. However, if it is a receive operation, then it stores its received value into some location local to the process.
- During the duplicated execution of the checkpointed piece, this end of communication is not executed anymore. However, if it is a receive operation, then it reads the previously received value from where it has been stored during the first execution of the checkpointed piece.



In this work we study checkpointing of MPI programs from a practical point of view. Before actually experimenting the refined receive-logging on a representative example, we want to study the question of the choice of checkpointed pieces. More precisely, we want to ask: if we have the choice between two checkpointed pieces, both of them reducing in the same way the peak memory consumption, is it better to choose the one that does not contain any orphan communication, in which case the application of the refined receive-logging implies the application of the standard approach? Or is it better to choose the one that contains a blend of orphan and non-orphan communications? One may think that the first choice is better as it does not require any message logging. However, in practice, this may not be right. In fact, applying checkpointing on MPI parallel programs has not only the cost of logging orphan receives, but also the cost of snapshots. In general the memory cost of the refined receive-logging may be written as:

$$CheckpointingCost = \sum_{i=1}^{n}(SnapshotCostP_i) + \sum_{i=1}^{m}(OrphanReceiveCost_i)$$

where $SnapshotCostP_i$ is the snapshot cost at each process P_i, n is the number of processes involved in the checkpointed piece, $OrphanReceiveCost_i$ is the memory cost of one orphan receive and m is the number of orphan receives inside the checkpointed piece. We note here that when the checkpointed piece includes many communications, it contains consequently few orphan receieves and thus the cost of $\sum_{i=1}^{m}(orphanReceiveCost_i)$ is small. However, including many communications, means also that sometimes we have many processes involved in the checkpointed piece and thus the cost of $\sum_{i=1}^{n}(SnapshotCostP_i)$ is high. Consider the example of figure 2 with two alternative checkpointed pieces C1 and C2. Piece C1 contains only non-orphan communications and piece C2 contains only orphan communications, actually only sends. We apply the refined receive-logging to both alternatives.



The cost of checkpointing piece C1 is $\sum_{i=1}^{3}(SnapshotCostP_i)$ and the cost of checkpointing piece C2 is $\sum_{i=0}^{3}(SnapshotCostP_i)$. We observe that although piece C2 contains many orphan communications, the memory cost of checkpointing C2 is lower than that of checkpointing C1. This can be explained by the fact that these orphan communications are sends and then they do not require any message logging. However, if the orphan communications were receives operations, the result might have been different. Actually, in this case we would have to compare the snapshot cost of the process 0 with the message logging cost of the orphan receives.

We validate our theorical results on a representative code, with various choices of checkpointed pieces in which we apply the refined receive-logging. We quantify the cost in term of memory and computation time for every resulting checkpointed adjoint.

References

[1]Dauvergne, B. and Hascoët, L. The Data-Flow Equations of Checkpointing in reverse Automatic Differentiation. *International Conference on Computational Science, ICCS 2006, Reading, UK*, 2006.

[2]Taftaf, A and Hascoët, L. On The Correct Application Of AD Checkpointing To Adjoint MPI-Parallel programs *European Congress on Computational Methods in Applied Sciences and Engineering*, 2016

Improved Fixed-Point Discrete Adjoint of simpleFoam

S. Akbarzadeh* and J-D. Müller†

School of Engineering and Materials Science (SEMS)

Queen Mary, University of London, London E1 4NS, United Kingdom

A. Sen‡ and M. Towara§ and U. Naumann¶

Software and Tools for Computational Engineering (STCE)

RWTH Aachen University, Aachen 52056, Germany

The implementation of a fixed-point iteration discrete adjoint algorithm for SIMPLE_type incompressible solvers[1] in OpenFOAM is presented. The algorithm reuses the operators and matrices of the linearised flow (primal) to build the adjoint solver that can be be "hot-started" within one-shot optimisation algorithms.

I. Introduction

The open source OpenFOAM (Open Source Field Operation and Manipulation) software is a CFD package that was first released in 2004.[2] The C++ language with object oriented programming is used to create all applications including solvers and utilities. The open source nature of package gives this flexibility to the users to develop and write their own applications. However, a solid understanding of OpenFOAM data structures in addition to the physics of problem is necessary. The package comes with preset solvers for a wide range of applications including SIMPLE_type incompressible solvers.

The well-known SIMPLE (Semi-Implicit Method for Pressure-Linked Equations)[3] algorithm has proved itself as an efficient incompressible solver in many applications and is used today in all major CFD codes like OpenFOAM. Using Picard iterations, the algorithm as a fixed-point iteration method, linearises and decouples the velocity and pressure in steady incompressible Navier-Stokes equations,

$$W^{n+1} = W^n + \mathbf{P} \cdot (\mathbf{b} - \mathbf{A}W^n), \tag{1}$$

where W is the vector of state variables (velocity and pressure), $\mathbf{A}$ is coefficient matrix and $\mathbf{P}$ is the pre-conditioner for the system arising from the SIMPLE scheme.

A discrete adjoint version of open-source fluid-dynamics package OpenFOAM has been presented by Towara et.al.[4] through application of dco/c++[5] operator overloading (**o-o**) AD tool. Moreover, the use of reverse accumulation for fixed point iteration resulted in performance improvement of the adjoint in steady-state problems.

II. Fixed-Point Discrete Adjoint of SIMPLE

Consider a scalar cost function, J, arising from determining a set of design variables, α, on boundary nodes of a CFD mesh. Any boundary deformation results in a change in flow state W which modifies the functional J. If a SIMPLE-type solver is used to compute the primal flow field, the brute-force reverse algorithmic differentiation (AD) results in

$$\bar{\alpha} = \frac{dX}{d\alpha}^T \bar{X}^T = \frac{dX}{d\alpha}^T \cdot \Big(\sum (\frac{\partial \mathbf{b}}{\partial X})^T \mathbf{P}^T \bar{W} - \sum (\frac{\partial \mathbf{A}}{\partial X})^T \mathbf{P}^T \bar{W} W^T\Big). \tag{2}$$

*SEMS, email:s.akbarzadeh@qmul.ac.uk.

†SEMS, email:j.mueller@qmul.ac.uk.

‡STCE, email:sen@stce.rwth-aachen.de.

§STCE, email:towara@stce.rwth-aachen.de.

¶STCE, email:naumann@stce.rwth-aachen.de.

The adjoint accumulation can be identified as[6]

$$v = \sum \mathbf{P}^T \bar{W}. \tag{3}$$

Using AD through **o-o**, the tape required for the building of the matrices can become prohibitive for larger meshes, even with linear solver treatment and checkpointing.[7]

When a steady-state primal is fully converged some savings can be achieved using reverse accumulation techniques[8] where only the last outer iteration of SIMPLE loop and its intermediate variables are needed for the derivative calculation. Moreover, computation of $\frac{\partial J}{\partial X}$, including $\frac{\partial \mathbf{b}}{\partial X}$ and $\frac{\partial \mathbf{A}}{\partial X}$, only coule be done once and do not need to be included in the fixed-point loop.

In one-shot optimisation[9] the state, adjoint and design problem converge concurrently. One of the main ingredients of this method is to have a fixed-point adjoint solver that can start from the solution obtained in the previous design step. Although, the reverse accumulation method is a fixed-point iteration but it always starts from scratch. The current author has derived a hand assembled fixed-point discrete adjoint formulation of SIMPLE,[1] based on the work of Stück et.al.,[10] which reuses the transpose of matrices and operators from converged primal SIMPLE loop,

$$v^{n+1} = v^n + \mathbf{P}^T \cdot (g - \mathbf{A}^T v^n) \tag{4}$$

where the right hand side g includes a number of terms: a) the derivative of the objective function J with respect to state variables; b) the adjoint advection (ADV) term and c) any explicit deferred corrections for higher order schemes. The terms need to be updated continuously during the adjoint iteration.

III. Implementation in OpenFOAM

In SIMPLE algorithm, the matrix coefficient, $\mathbf{A}$, includes the convection-diffusion operator, $\mathbf{F}$, and its Schur-Complement approximation for momentum and pressure equations, respectively. When the primal converges, these operators are linearised and in the fixed-point discrete adjoint, we reuse their transpose on the left hand side of system. On the other hand, the ADV term on the right hand side requires the derivative of $\mathbf{F}$ and boundary terms, $\mathbf{b}$, with respect to state variables ($\frac{\partial \mathbf{F}}{\partial \mathbf{W}}$, $\frac{\partial \mathbf{b}}{\partial \mathbf{W}}$). But due to the nature of reverse AD by **o-o** tools, in large cases the memory footprint needed to compute these derivatives is pretty huge. The aim of this work is to implement a fixed-point discrete adjoint of simpleFoam solver for first order upwind scheme and hand differentiate the internal ADV term whilst the boundary terms are differentiated by AD.

This study is designed as a proof-of-concept to check: a) whether this implementation is feasible by dco/c++ for SimpleFoam, and/or b) potential of other possibilities like 'light-weight' source-transformation of C++ embedded in **o-o** code.

References

[1] Akbarzadeh, S., Wang, Y., and Mueller, J. D., chap. Fixed point discrete adjoint of SIMPLE-like solvers, AIAA Aviation, American Institute of Aeronautics and Astronautics, Jun 2015.

[2] "OpenFOAM," http://www.openfoam.com/.

[3] Patankar, S. V. and Spalding, D., "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flow," *Int. J. of Heat and Mass Transfer*, Vol. 15, No. 10, 1972, pp. 1787–1806.

[4] Towara, M. and Naumann, U., "A Discrete Adjoint Model for OpenFOAM," *Procedia Computer Science*, Vol. 18, 2013, pp. 429 – 438, 2013 International Conference on Computational Science.

[5] Lotz, J., Leppkes, K., and Naumann, U., "dco/c++ efficient derivative code by overloading in c++," Tech. rep. aib-2011-05, RWTH Aachen, 2011, http://aib.informatik.rwth-aachen.de/2011/2011-05.pdf.

[6] Giles, M., "On the Iterative Solution of Adjoint Equations," *Automatic Differentiation of Algorithms*, edited by G. Corliss, C. Faure, A. Griewank, L. Hascoët, and U. Naumann, Springer-Verlag New York, Inc., New York, NY, USA, 2002, pp. 145–151.

[7] Griewank, A. and Walther, A., *Evaluating Derivatives: Principles and Techniques of Algorithmic Differentiation*, SIAM, Philadelphia, PA, second edition ed., 2008.

[8] Christianson, B., "Reverse accumulation and attractive fixed points," *Optimization Methods and Software*, Vol. 3, No. 4, 1994, pp. 311–326.

[9] Ta'asan, S., "One Shot methods for Optimal Control of Distributed Parameter Systems." *Finite dimensional control. Report 91-2, NASA Contractor Report 187497, ICASE*, 1991.

[10] Stück, A. and Rung, T., "Adjoint complement to viscous finite-volume pressure-correction methods," *Journal of Computer Physics*, Vol. 248, 2013, pp. 402 – 419.

Probabilistic analysis of topologically optimized structures considering geometric imperfections

Julian K. Lüdeker[*] and Benedikt Kriegesmann[†]

Hamburg University of Technology, Am Schwarzenberg-Campus 4, 21073 Hamburg, Germany

I.Introduction

THE industrialization of additive layer manufacturing has raised the intention of aircraft industry for topology optimization, as it allows manufacturing arbitrary shapes of structures. Standard topology optimization using the SIMP approach minimizes the compliance for a given volume fraction.[1] Several works enhanced topology optimization for considering stress constraints,[2,3] as this is more relevant for practical application that the stiffness. However, these approaches do not consider the sensitivity with respect to uncertainties and therefore provide designs with little robustness.

Kharmanda et al. suggested combining reliability based design optimization with topology optimization, which is referred to as reliability-based topology optimization (RBTO).[4] Other studies followed this idea and confirmed the finding that the design obtained from RBTO can differ significantly from the one obtained using classical topology optimization.[5,6] In these publications only parameters were considered as random which are independent of the design, such as Young's modulus, load magnitude and the size of the design space. The scatter of important measured like material strength, defects and geometry was not considered. Especially geometrical deviations are challenging to be considered as random within the RBTO, but it is known to have an effect on stability and fatigue performance.

The present work suggests a procedure to account for geometric deviations from the nominal structure in probabilistic analyses. First, a procedure is presented to describe the result of a topology optimization by Non-Uniform Rational B-Splines (NURBS). Secondly, the utilization of the NURBS representation for probabilistic analyses is demonstrated.

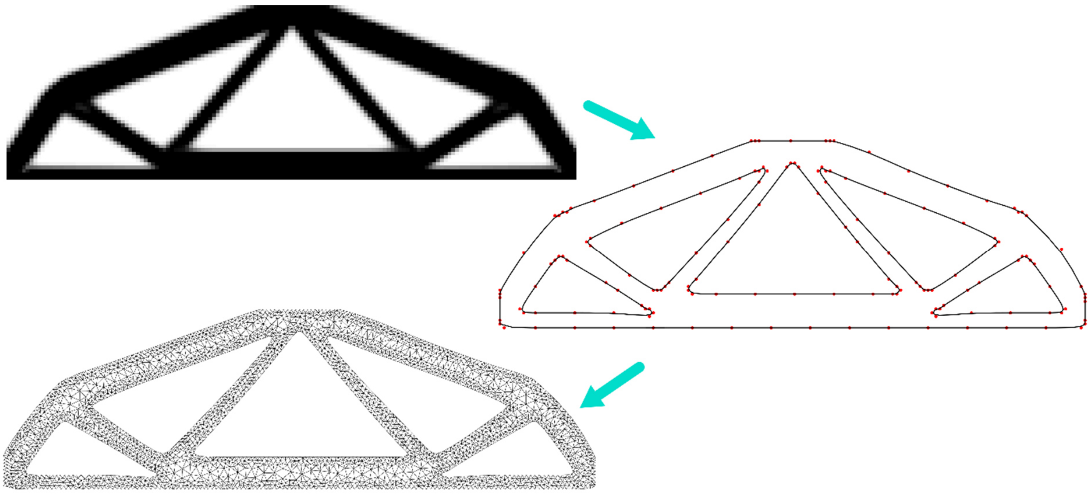


Figure 1. Result of a topology optimization (top), an automatically generated geometric interpretation (middle) and the finite element mesh (bottom).

II.Parameterization of Topologically Optimized Geometry

Topology optimization using the SIMP approach provides a distribution of normalized densities, which vary between 0 (no material) and 1 (fully solid). Densities in between 0 and 1 can be interpreted as the homogenized property of a somehow porous cell. However, in practice often a choice is made to consider or not a cell as part of the structure by choosing a density threshold, especially since areas with densities in between 0 and 1 are located at the edge of the final structure (see Figure 1, top).

[*] Scientific assistant, Working group New Design and Sizing Methods for Hybrid Airframe Structures.

[†] Juniorprofessor, Working group New Design and Sizing Methods for Hybrid Airframe Structures.

For the current work, the objective was to define the edge independent of an engineer's choice. This is achieved by defining the edge of the structure along the lines of maximal gradients of the density fields obtained from topology optimization. The representation of the edge shall be parameterized such that its parameters can be used for further shape optimization and probabilistic analyses. Therefore, NURBS have been used to describe the structure edges (see Figure 1, middle), which allows adjusting the geometry by modification of the control points. (For details on NURBS, see e.g. Ref. [7].) The control points of NURBS (red dots in Figure 1, middle) are concentrated at corners of the geometry. From the NURBS representation, a Finite Element model is generated (see Figure 1, bottom), which is analyzed with respect to stress distribution.

III.Probabilistic analysis

A probabilistic analysis is performed considering geometric imperfection as random input and the maximal occurring stress as output parameter. The scatter of geometry is described by the coordinates of the control points of the NURBS $x_{c,i}$ and $y_{c,i}$, which are summarized in the random vector **X**.

$$\mathbf{x} = \left(x_{c,1}, y_{c,1}, \ldots, x_{c,i}, y_{c,i}, \ldots, x_{c,p}, y_{c,p}\right)^T \tag{1}$$

Given a sample of manufactured specimens, the control point coordinates are adjusted to describe the geometry of the specimen. Hence, a realization $\mathbf{x}^{(j)}$ of the random vector **X** is obtained for each specimen. From the realizations, the mean vector **μ** and the covariance matrix **Σ** of random vector **X** are estimated. These are used for transforming the random vector **X** to the random vector **Z**, by

$$\mathbf{X} = \mathbf{Q}\,\mathbf{D}^{\frac{1}{2}}\,\mathbf{Z} + \boldsymbol{\mu} \qquad \boldsymbol{\Sigma} = \mathbf{Q}\,\mathbf{D}\,\mathbf{Q}^T \;. \tag{2}$$

The matrixes **Q** and **D** are obtained from the spectral decomposition of **Σ**. The random vector **Z** has uncorrelated entries with a mean value of 0 and a standard deviation of 1. Furthermore, the length of **Z** cannot exceed the number of measurements used to build the covariance matrix. Therefore, this transformation significantly reduces the number of parameters required to describe geometric imperfections. (For more details, see e.g. Ref. [8].) In the present case, $p = 158$ control points are used and **X** has a length of 316. Since 20 specimens are considered to build **μ** and **Σ**, **Z** has a length of only 19.

The random vector **Z** is considered to generate samples in a Monte Carlo simulation as well as for preforming first-order second-moment analysis,[8] which requires estimating the gradient of the probabilistic objective function (here, maximum stress) with respect to scattering parameters. The combination of the NURBS representation with the transformation (2) allows describing the scatter of a complex shape with very few parameters and therefore allows for very efficient probabilistic analysis.

For the present work, no measurements of manufactured specimen are available. Therefore, virtual specimens have been generated by perturbation the initial geometry. The subsequent steps are the same, but an experimental validation of the procedure is pending.

IV.Conclusion and Outlook

An efficient procedure is presented to capture the randomness of geometric imperfections for probabilistic analyses of topologically optimized structures. The procedure is easy to integrate into a reliability based shape optimization of complex structures.

References

1 Bendsøe, M. P., and Kikuchi, N., "Generating optimal topologies in structural design using a homogenization method," *Computer Methods in Applied Mechanics and Engineering*, vol. 71, Nov. 1988, pp. 197–224.

2 Duysinx, P., and Sigmund, O., "New developments in handling stress constraints in optimal material distribution," *American Institute of Aeronautics and Astronautics*, vol. Paper 98-4906, 1998, p. 10.

3 Holmberg, E., Torstenfelt, B., and Klarbring, A., "Stress constrained topology optimization," *Struct Multidisc Optim*, vol. 48, 2013, pp. 33–47.

4 Kharmanda, G., Olhoff, N., Mohamed, A., and Lemaire, M., "Reliability-based topology optimization," *Structural and Multidisciplinary Optimization*, vol. 26, Mar. 2004, pp. 295–307.

5 Jung, H.-S., and Cho, S., "Reliability-based topology optimization of geometrically nonlinear structures with loading and material uncertainties," *Finite Elements in Analysis and Design*, vol. 41, Dec. 2004, pp. 311–331.

6 López, C., Baldomir, A., and Hernández, S., "Deterministic versus reliability-based topology optimization of aeronautical structures," *Structural and Multidisciplinary Optimization*, Nov. 2015.

7 Farin, G. E., *Curves and surfaces for CAGD a practical guide*, San Francisco: Morgan Kaufmann, 2002.

8 Kriegesmann, B., "Probabilistic Design of Thin-Walled Fiber Composite Structures," Mitteilungen des Instituts für Statik und Dynamik der Leibniz Universität Hannover 15/2012, ISSN 1862-4650, Gottfried Wilhelm Leibniz Universität Hannover, 2012.

Optimisation for scaling up of plywood sandwich panels with rigid PU foam-cores

Kaspars Kalnins[*] and Gints Jekabsons[†]
Riga Technical University, Institute of Materials and Structures, Riga, Azenes st. 16/20, LV1048, Latvia

Edgars Labans[‡]
Delft University of Technology, Faculty Aerospace Engineering, Delft ,Kluyverweg 1, 2629 HS, The Netherlands

Current paper deals with numerical analysis and optimisation with aim of scaling up of the mechanical and thermal behavior of rib-stiffened sandwich panels with plywood and PU foam core constituents. The effect of the skin and rib thicknesses and core density on mechanical and thermal properties has been analysed. Sandwich panel stiffness and of effective thermal conductivity were acquired by means of numerical models in ANSYS software. Parametrical optimisation of the cross section dimensions and material properties was performed to found the best trade-off between stiffness, structural weight and thermal properties. Comparing optimised sandwich structures with tradition plywood boards it is possible to found equivalent stiffness sandwich panels with weight reduction up to 35 % and effective thermal conductivity of 0.029 W/m*k (reference to 0.12 for solid plywood board).

I. Introduction

Wood based structural materials like timber beams, plywood and oriented strand board (OSB) are widely used in construction of housing and industrial building. Common practice of building large span load bearing floors and roofs currently heavy relies on assembly procedures inside construction site. In order to improve assembly time, quality and cost efficiency pre-fabricated wood based sandwich panels could be used instead. Thus several engineering challenges should be faced in designing of market oriented wood based sandwich panel.

Considering that there is several design variables for sandwich panel cross-section, the optimization allows to track the most efficient combinations of these variables. In addition to optimization of mechanical performance and mass, additional aspects like thermal/sound insulation and material/assembly cost should be addressed.

Advantages of stiffness and weight optimisation for sandwich panels with weak foam cores are described in several research articles[1,2] . Optimisation of the rib-stiffened panels without any core filler is given in previous research by Labans and Kalnins[3] where clear weight saving of more than 60 % comparing with reference plywood boards has been achieved. In additional optimisation results were experimentally validated by making 4-point bending tests on panel prototypes. This research was further extended by adding third optimization objective of thermal conductivity [4]. Novel contribution to design and optimisation of plywood based sandwich panels also has been provided in several recent articles[5,6].

In current research optimization of large scale (up to 4m) of wood based sandwich panels with plywood surfaces, stiffeners and natural foam core has been performed to find the most beneficial design regarding structural weight and cost efficiency. Manufacturing constraints, baseline of mechanical and thermal insulation performance also has been included in optimization constraints.

II.Methods and tools

Mechanical and thermal responses have been acquired by the means of numerical models based on Finite Element Method (FEM). Commercial software ANSYS has been employed for this purposes. Combined shell and

Communicating Author: kaspars.kalnins@sigmanet.lv
[*]Dr.sc.ing. Leading researcher, Institute of Materials and Structures
[†] Dr.sc.ing. Researcher, Institute of Materials and Structures.
[‡] Dr.sc.ing. Researcher, Delft University of Technology

solid element types have been combined for stiffness calculations with 4-node SHELL181 elements and 8-node SOLID185 elements. Thermal model of the cross-section numerically represented in 2D model with PLANE55 elements. Area load of 1.5 KPa applied on the sandwich surface corresponds to typical snow load magnitude in Latvia. Length of the panel has been fixed to 4 m – longest available plywood sheet size at local manufacturer. Restrictions on deflection and thermal conductivity values taken according Euro Code for timber structures.

The cross section of a corrugate panel has been characterised with five design variables (Table 1). Separate parameter assigned for core density P5, which has linear relation with foam mechanical properties. Sequential space filling design based on Latin Hypercube with Means Square error criterion has been evaluated. Response surface method used in approximation of optimisation responses.

Table 1. Design variables

Parameter	Lower bound	Upper bound	Step	Units
Number of surface plies - P1	5	13	2	-
Number of stiffener plies –P2	5	13	2	-
Stiffeners distance– P3	50	300	50	mm
Total section height – P4	80	200	-	mm
Foam E-modulus - P5	75	300	-	MPa

III.Main results

As the result of parametric optimization several feasible designs of wood based sandwich panels has been elaborated. Most effective lightweight sandwich panel could be made applying birch plywood with 9 mm thickness for skins and stiffeners. Distance between stiffeners should be set to 200 mm and overall thickness of the panel at 150 mm, to fulfill thermal insulation requirements with foam filler density of 40 kg/m^3. Such design pass deflection limitation of 1/300 of span length and thermal insulation requirements where overall heat transfer coefficient < 0.23 W / m^{2}K. Cost saving for this type of sandwich panels could be reached mainly by substituting skin and/or stiffener material by cheaper alternative like OSB or high density fiber board(HDF) sheets. Although additional analysis should be applied to measure increase of manufacturing costs by number of material types applied.

Acknowledgements

The research has received funding from Latvia state research program under grant agreement "Innovative and multifunctional composite materials for sustainable buildings (IMATEH)".

References

[1]Kawasaki, T., Zhang, M., Wang, Q., Komatsu, K. & Kawai, S., "Elastic moduli and stiffness optimization in four-point bending of wood-based sandwich panel for use as structural insulated walls and floors", Journal of Wood Science, vol. 52, no. 4, 2006, pp. 302-310.

[2]Fernandez-Cabo J.L., Majano-Majano A., San-Salvador Ageo L., Ávila-Nieto M. Development of a novel façade sandwich panel with low-density wood fibres core and wood-based panels as faces// European Journal of Wood and Wood Products, vol. 69, no. 3, 2011, pp. 459-470.

[3]Labans, E., Kalniņš, K. Experimental Validation of the Stiffness Optimisation for Plywood Sandwich Panels with Rib-Stiffened Core. Wood Research, 2014, Vol.59, Iss.4, pp.793-802.

[4]Labans, E., Kalniņš, K., Zudrags, K., Rudzīte, S., Kirpluks, M., Cabulis, U. Evaluation of plywood sandwich panels with rigid PU foam-cores and various configurations of stiffeners. In: Proceedings of the 3rd International conference on Optimization and Analysis of Structures, Estonia, Tartu, 23-25 August, 2015, pp.45-51.

[5]Sliseris, J., Frolovs, G., Rocens, K. & Goremikins, V."Optimal design of GFRP-plywood variable stiffness plate", Procedia Engineering, 2013, ,pp. 1060.

[6]Banerjee, S., Bhattacharyya, D., 2011: Optimal design of sandwich panels made of wood veneer hollow cores. Composites Sci. Technol. 71(4): 425-432

Sequential element rejection and admission method (SERA) for topology optimization using a constraint on perimeter

Ruben Ansola, Alain Garaigordobil, Estrella Veguería
Mechanical Engineering Department, School of Engineering, University of The Basque Country (UPV-EHU), Alda. Urquijo, s/n, 48013 Bilbao, Spain

Osvaldo M. Querin
School of Mechanical Engineering, University of Leeds, Leeds LS2 9JT, UK.

This paper presents an implementation of the perimeter constraint for topology optimization of elastic structures using the Sequential Element Rejection and Admission Method (SERA). The perimeter constraint allows the designer to control the number of holes in the optimal design and to establish their characteristic length scale. This work shows an improved SERA methodology incorporating a strategy to efficiently control the optimization process satisfying a constraint on structural perimeter.

Topology optimization is a computational approach that optimizes material distribution within a fixed design domain and for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of the design requirements. It is an expanding research field of computational mechanics which has been growing very rapidly and has attracted the interest of numerous applied mathematicians and engineering designers, becoming extremely popular in the last years. Recently, additive manufacturing has opened the possibility to overcome limits currently imposed by conventional manufacturing techniques. In this sense, controlling the structural complexity for topology optimization problems has become an important investigation field since it affects the number and cost of the scaffold structures required by some of these manufacturing processes. It will be shown that the SERA method can be efficiently combined with the perimeter control method and it is capable of suppressing mesh dependency and control the number of holes and members of the final solution.

Since the landmark paper of Bendsoe and Kikuchi [1], where a so-called microstructure or homogenization based approach was used, numerical methods for topology optimization have been investigated extensively. At present the most popular topology optimization method is the SIMP method, which stands for “Solid Isotropic Material with Penalization”, proposed in the late eighties by Bendsoe [2]. A well known problem associated with topology optimization is that the optimal solution depends on the discretization level, as observed in many applications based on the finite element method. In order to ensure existence of solutions to the topology optimization problem, some sort of restriction on the resulting design must be introduced, combining the power law approach with, e.g., a perimeter constraint [3]. During last years Level Set Methods have emerged as an attractive and promising alternative to perform structural shape and topology optimization, inspired in the work on topological derivatives by Sokolowski and Zochowski and the paper by Sethian and Wiegman [4]. Apart from above mentioned approaches, a number of heuristic or intuition based approaches have effectively addressed a variety of size, shape and topology optimization problems. An important branch of these approaches for topology optimization is the evolutionary structural optimization approach (ESO) by Xie and Steven. The initial concept was that by systematically removing inefficient materials (elements with lowest strain energy density), the structure evolves towards an optimum. Its application in topology optimization of continuum media is quite extensive, see e.g., Xie and Steven [5]. Although initially solely based on intuition, this basic idea has developed from simple hard-kill strategies to more efficient soft-kill bi-directional schemes (BESO), which allow efficient materials to be added in addition to the inefficient ones being removed [6]. The newer BESO method has demonstrated its strength in solving a variety of topology optimization problems, but as it is presently defined, it uses a power law (SIMP) parametrization strategy and standard filtering techniques similar to those used in the density approach in order to stabilize results, so it could be categorized as a discrete update version of the standard SIMP scheme. Rozvany and Querin proposed some improvements of this method under the term SERA (Sequential Element Rejection and Admission) where a “virtual material” was introduced, without the use of any intermediate densities or power law interpolations [7]. Additionally, two separate criteria are considered in the topology optimization process by SERA method, where the sensitivity numbers of real and virtual material present in the domain are sorted out separately. These ideas were developed for fully stressed design and extended to most of the classical problems in structural topology optimization and compliant mechanisms design [8,9].

The proposed perimeter control algorithm analyzes the effect on this constraint when an element is removed or added through the SERA optimization process, since it has different effects on the structural perimeter depending on its current connections with the neighboring elements. The classical algorithm based on a continuous approximation of the perimeter is substituted here by a discrete implementation of the algorithm, without the use of any intermediate densities or power law interpolations necessity. Preliminary results show the capability of SERA method in combination with perimeter control and are be demonstrated through different numerical examples. The following picture shows some preliminary results of the well known MBB beam problem when the SERA method is applied in combination with the perimeter control developed in this work.

Figure1 : MBB optimum topology with high and low perimeter constraints

Acknowledgements

We gratefully acknowledge the financial support from the Ministry of Economy and Competitiveness of Spain under the project DPI2015-64863-R.

References

[1] Bendsoe M.P., Kikuchi N., “Generating optimal topologies in structural design using a homogenization method”, Comput Methods Appl Mech Eng Vol. 71, pp. 197–224, 1988.

[2] Bendsoe M.P., “Optimal shape design as a material distribution problem”, Struct Optim Vol. 1, pp. 193–202, 1989.

[3] Haber R.B., Bendsoe M.P., Jog C.S., “A new approach to variable topology shape design using constraint on the perimeter”, Struct Optim Vol. 11, pp. 1–12, 1996.

[4] Sethian J.A., Wiegman A., “Structural boundary design via level set and immersed interface methods”, J Comp Phys Vol. 163, pp. 89–528 (2000).

[5] Xie Y.M., Steven G.P., “Evolutionary structural optimization Evolutionary structural optimization”, Berlin, Heidelberg, New York: Springer-Verlag, 1997.

[6] Querin, O.M., Evolutionary structural optimization: stress based formulation and implementation. Ph.D. Thesis, Department of Aeronautical Engineering, University of Sydney, Australia, 1997.

[7] Rozvany G.I.N, Querin O, “Sequential element rejections and admissions (SERA) method: applications to multiconstraint problems”, Proceedings of the 10th AIAA/ISSMO Multidisc. Anal. Optim. Conference, Albany, NY, Aug.–Sept. 2004.

[8] Alonso C, Querin O.M., Ansola R., “A Sequential Element Rejection and Admission (SERA) method for compliant mechanisms design”, Struct Optim Vol. 47, pp. 795–807, 2013.

[9] Alonso C, Ansola R, Querin O.M., “Topology synthesis of multi-material compliant mechanisms with a Sequential Element Rejection and Admission method”, Finite Elem Anal Des Vol. 85, 11–19, 2014.

Temporal and spatial checkpoint coarsening in adjoint shape optimisation for unsteady flow

Jan Hückelheim* and Jens-Dominik Müller†
Queen Mary University of London, London, W2 3RA, United Kingdom

I. Introduction

Gradients are a key ingredient in optimisation, uncertainty quantification or flow control[1] and can be computed efficiently using the adjoint method.[2] Adjoint gradients of flow problems can be computed by discretising the adjoint flow equations (leading to the *continuous adjoint* approach), or by differentiating a flow solver using techniques such as algorithmic differentiation (the *discrete adjoint* approach).

Unsteady flow problems pose challenges to continuous and discrete adjoint methods. The memory requirements become prohibitive as discussed in II, since the full flow trajectory needs to be known to compute the adjoint field. In addition, chaotic behaviour in the flow naturally leads to divergent intermediate values in the adjoint field. This is an inherent property of the sensitivities of chaotic flow. Other authors have presented methods such as least-squares shadowing[3] to obtain a well-defined adjoint solution in these cases, but with a runtime and memory footprint that is infeasible for large applications.

In previous work, we presented checkpointing with time gaps as an approach to reduce the memory requirements of unsteady adjoint methods while retaining most of the solution accuracy.[4] The method was shown to be cheap and yield good compression ratios at a reasonable accuracy, but had stability problems for high ratios. We further demonstrated spatial checkpoint coarsening based on a geometric multigrid sequence,[5] which was effective in improving the stability of adjoints for chaotic flow.

In this work we improve both the spatial and temporal coarsening methods and combe them on a more challenging test case. The spatial coarsening method is extended by considering a multigrid sequence that is tailor-made for incomplete checkpointing, in contrast to our previous work where we used a standard cell-collapsing scheme with semi-coarsening. The temporal coarsening is extended by considering time-averaged checkpoints instead of the snapshots that we used in the previous work. With this we stabilise the method. The effectiveness of these approaches is shown on a U-shaped internal turbine blade cooling passage[6–8] simulated with implicit LES, which was shown to be more appropriate than RANS for this test case.[9] The test case has been used by other authors for steady adjoint-based shape optimisation,[8] the flow has been studied in simulations[6] and experiments.[7]

II. Unsteady adjoint and coarse checkpointing

The viscous unsteady flow equations, including the DES or LES turbulence model, can be written as

$$\frac{\partial U}{\partial t} + R(U) = 0,$$

and discretised in space and time and solved forward in time. The unsteady adjoint system reads

$$-\frac{\partial v}{\partial t} + \left(\frac{\partial R}{\partial U}\right)^T v - \left(\frac{\partial J}{\partial U}\right)^T = 0$$

and can be discretised in time by using the same operators as in the primal case, and in space by using either the continuous or discrete adjoint approach. The system is solved backwards in time, where the solution of the adjoint equation at each time requires the primal flow field at that time step. If the memory is insufficient to store the flow trajectory, checkpointing[13] is commonly used to recompute the transient flow field starting from selected stored time steps. An alternative that is discussed in the literature is checkpoint compression.[14] However, both approaches increase the computational effort.

*PhD candidate, School for Engineering and Materials Science, j.c.hueckelheim@qmul.ac.uk
†Senior lecturer, School for Engineering and Materials Science, j.mueller@qmul.ac.uk

We use a CFD solver with implicit LES on unstructured grids and dual timestepping using an implicit solver to converge the inner iterations as presented in.[10] The adjoint solver is generated using the automatic differentiation tool Tapenade[11] with some hand-coded optimisations for improved speed.[12]

The routine to store checkpoints in the flow solver is augmented by using the standard multigrid restriction operator to transfer the solution to the coarse mesh. The approximate solution is restored during the adjoint computation using the multigrid prolongation operator. The temporal coarsening is performed by combining several time steps to form an average. During the adjoint computation, we use linear interpolation to blend from one averaged snapshot to another.

III. Results

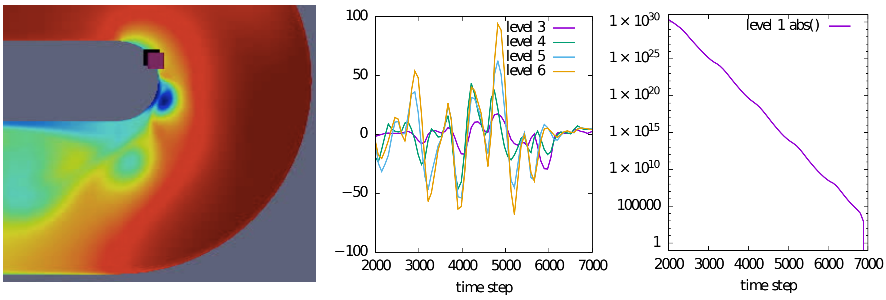


Figure 1: **Left:** LES flow and location of flow control. **Middle:** Sensitivity of pressure loss to flow control using spatial checkpoint coarsening. **Right:** Divergence of sensitivities without checkpoint coarsening

Acknowledgements

This work has been conducted within the About Flow project on "Adjoint-based optimisation of industrial and unsteady flows", which has received funding from the European Union's Seventh Framework Programme for research, technological development and demonstration, Grant Agreement No. 317006.

References

[1] Byung Joon Lee and Meng-Sing Liou. Unsteady adjoint approach for design optimization of flapping airfoils. *AIAA Journal*, 50(11):2460–2475, 2015/05/03 2012.

[2] M. Giles, N. Pierce, M. Giles, and N. Pierce. *Adjoint equations in CFD - Duality, boundary conditions and solution behaviour*. American Institute of Aeronautics and Astronautics, 2015/05/03 1997.

[3] Qiqi Wang, Rui Hu, and Patrick Blonigan. Least squares shadowing sensitivity analysis of chaotic limit cycle oscillations. *Journal of Computational Physics*, 267(0):210 – 224, 2014.

[4] Jan Hueckelheim and Jens-Dominik Müller. Checkpointing with time gaps for unsteady adjoint CFD. In *EUROGEN-2015*, volume 11, September 2015.

[5] Jan Hueckelheim and Jens-Dominik Müller. Checkpointing with time gaps for unsteady adjoint CFD. In *ISROMAC-2016*, 2016.

[6] Tom Verstraete, Filippo Coletti, Jérémy Bulle, Timothée Vanderwielen, and Tony Arts. Optimization of a U-Bend for minimal pressure loss in internal cooling channelsópart I: Numerical method. *Journal of Turbomachinery*, 135(5):051015, 2013.

[7] Filippo Coletti, Tom Verstraete, Jérémy Bulle, Timothée Van der Wielen, Nicolas Van den Berge, and Tony Arts. Optimization of a U-Bend for minimal pressure loss in internal cooling channelsópart II: Experimental validation. *Journal of Turbomachinery*, 135(5):051016, 2013.

[8] Sebastian Willeke and Tom Verstraete. Adjoint optimization of an internal cooling channel U-Bend. In *ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, pages V05AT11A029–V05AT11A029. American Society of Mechanical Engineers, 2015.

[9] Rémy Fransen. *LES based aerothermal modeling of turbine blade cooling systems*. PhD thesis, Institut National Polytechnique de Toulouse-INPT, 2013.

[10] Shenren Xu, David Radford, Marcus Meyer, and Jens-Dominik Müller. Stabilisation of discrete steady adjoint solvers. *Journal of Computational Physics*, (accepted for publication):–, 2015.

[11] Laurent Hascoët and Valérie Pascual. The tapenade automatic differentiation tool: principles, model, and specification. Research Report RR-7957, May 2012.

[12] Faidon Christakopoulos, Dominic Jones, and Jens-Dominik Müller. Pseudo-timestepping and verification for automatic differentiation derived CFD codes. *Computers Fluids*, 46(1):174 – 179, 2011.

[13] Andreas Griewank and Andrea Walther. Algorithm 799: Revolve: An implementation of checkpointing for the reverse or adjoint mode of computational differentiation. *ACM Trans. Math. Softw.*, 26(1):19–45, March 2000.

[14] Eric C. Cyr Tim Wildey and John Shadid. Adjoint based a posteriori error estimates using data compression. In C. Tiago J. P. Moitinho de Almeida, P. D ez and N. Parés, editors, *VI International Conference on Adaptive Modeling and Simulation*, 2013.

Aerodynamic Shape Optimization Using the Adjoint-based Truncated Newton Method

Mehdi Ghavami Nejad
Evangelos M. Papoutsis-Kiachagias
Kyriakos C. Giannakoglou
National Technical University of Athens, School of Mech. Eng., Parallel CFD & Optimization Unit , Athens, Greece

This paper presents the development and application of the truncated Newton (TN) method in aerodynamic shape optimization problems. The method has been developed in OpenFOAM for incompressible laminar flows with the aim of showing its advantages over standard gradient-based optimization algorithms. Generalized minimal residual (GMRES) is used for solving the Newton equations. GMRES only requires the computation of the product of the Hessian of the objective function and a vector which enables us to refrain from computation of the Hessian itself; computational cost of the latter scales with the number of design variables and becomes unaffordable in large-scale problems with numerous design variables. A combination of the continuous adjoint method and direct differentiation is used for computation of all Hessian-vector products. Also, a grid displacement PDE (Laplace equation) is used for computing necessary derivatives of grid displacement w.r.t the design variables. The optimization of a 3D S-bend duct provided as one of the test cases of the AboutFLOW ITN programme is presented with the aim of minimizing the total pressure losses.

I. Introduction

In aerodynamic shape optimization, the adjoint approach is widely used for computation of the objective function gradient w.r.t the design variables since the computational cost is independent of the number of the design variables. Moreover, Newton optimization algorithms might lead to faster convergence (at least, in terms of optimization cycles) compared to conventional optimization algorithms solely relying on the gradient. However, the computational cost of the Hessian matrix required for Newton method scales with the number of design variables which is the main reason that the exact Newton method is limited to applications with few design parameters. One alternative could be the exact initialization of the quasi Newton method in which the exact Hessian is only computed in the first optimization cycle and then, it is approximately updated[1]. Although this method could be more efficient than the Newton method, computation of the exact Hessian, even for once, might become prohibitive in large cases with numerous design variables. In these cases, TN can be used instead.

II. Development of the TN method

Assuming F is a general objective function, the Newton method accelerates the optimization algorithm in which the design variables b_i ($i = 1,\ldots,N$) are updated by solving the Newton's system :

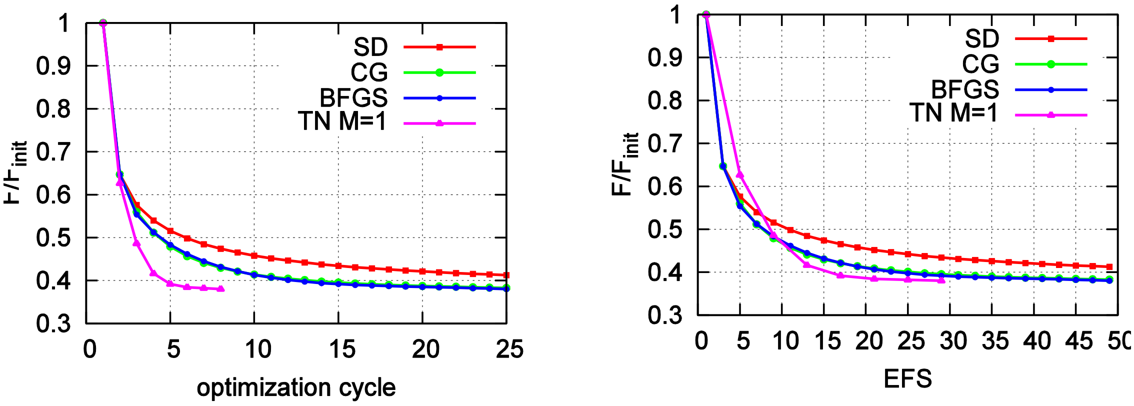
$$\frac{\delta^2 F}{\delta b_i \delta b_j} \Delta b_j = -\frac{\delta F}{\delta b_i} \qquad (1)$$

$$b_j^{new} = b_j^{old} + \Delta b_j \quad j = 1, \ldots, N$$

The r.h.s of equation (1) can be computed via the adjoint method since it is nothing more than the gradient of F. Although, in theory, Hessian is symmetric and any iterative algorithm which only relies on the computation of the Hessian-vector products without requiring the knowledge of the Hessian itself should be appropriate for solving equation (1), this is not the case in CFD applications. The Hessian expression which is obtained through the AV-DD approach (i.e., use of adjoint method for computation of $\frac{\delta F}{\delta b_i}$ and direct differentiation for computation of variations of the adjoint and primal fields) becomes symmetric if all equations are converged to machine accuracy[2]. In CFD-based optimizations, it is quite common not to converge primal and adjoint equations to machine accuracy in each optimization cycles in order to reduce the total CPU cost. This can deteriorate the symmetry of the Hessian matrix which renders algorithms like conjugate gradient (CG) inappropriate for solving the equation (1). Hence, GMRES which is for non-symmetric systems is used instead.

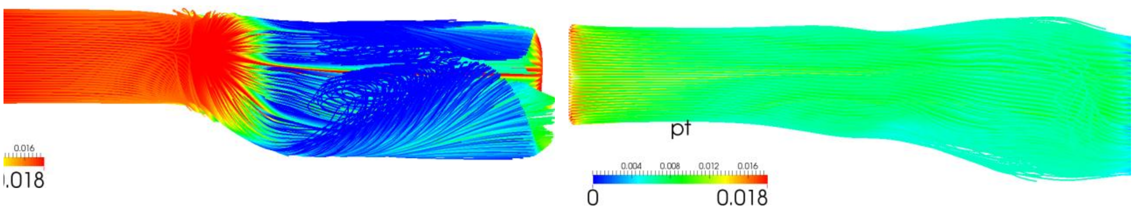
III. Results

In this section, the results of optimizing a 3D S-bend duct with the aim of minimizing the total pressure losses are presented. The flow is laminar with a Reynolds number of Re = 400 based on the inlet hydraulic diameter and the mesh comprises 474000 hexahedra. A 9 × 7 × 9 control grid is used to parameterize part of the duct which results in 375 design variables. In figure 1, the convergence history of the developed TN algorithm is compared to those of steepest descent (SD), conjugate gradient (CG) and BFGS methods. It can be observed that TN outperforms other methods in terms of both the number of optimization cycles and the equivalent flow solutions (EFS). In figure 2, the flow streamlines on the reference and optimized shape are compared indicating the significant reduction of the flow recirculation which in turn leads to a 60% decrease in the total pressure losses.



(a) **(b)**

Figure 1. Comparison of the convergence of SD, CG, BFGS and TN w.r.t (a) optimization cycles (b) EFS



(a) **(b)**

Figure 2. Velocity stream lines plotted for (a) reference shape, (b) optimized shape. Streamlines are colored based on the total pressure values. The intense flow recirculation close to the bottom side of the wall has drastically been reduced leading to a 60% decrease in the objective function.

References

[1]Zervogiannis, T., Papadimitriou, DI., and Giannakoglou, KC., "Total Pressure Losses Minimization in Turbomachinery Cascades Using the Exact Hessian," *Computer Methods in Applied Mechanic and Engineering*, Vol. 199, No. 41, 2010, pp. 2697, 2708.

Adjoint-Based Numerical Error Estimation

Sheikh Razibul Islam*

Warsaw University of Technology, Warsaw, 00-665, Poland

Jacek Szumbarski†

Warsaw University of Technology, Warsaw, 00-665, Poland

Estimations of errors are one of the critical pointers while improving the fidelity and correctness of CFD program. Convection dominated numerical simulations of aerodynamic flows are largely affected by spatial discretization. Importance of generating appropriate mesh for such CFD simulations demands special attentions, as improper resolution at area which affect the output of interest, can have severe consequence. It is in this predicament, output-based adaptation methodology, which take into account the error escalation effects, can be advantageous. The additional implementation cost for adjoint solver is trivial compared to its reduced computational cost. Error estimation and adaptive indicators in a specific solution for a functional output can be determined by adjoint method. This article focuses on the recent advancements on error estimation techniques in computational fluid dynamics. In this paper, adjoint methodology is used to extend error estimation in which the true error is not employed rather output error is approximated as an inner product of the residuals and the adjoint variables associated with the functional output. Adjoint solver has been developed using the in house CFD code RED which is based on Residual distribution Scheme. Algorithmic differentiation tool dco/c++ is used to implement the adjont code. Additionally, output error based mesh adaptation techniques are also reviewed.

I. Introduction

The use of adjoint-based CFD simulation is becoming increasingly prevalent in the design process within the industry. Continuing developments in both computing technology and algorithmic development are ultimately leading to attempts at simulating ever-larger, more complex problems. The efficiency of the underlying numerical algorithms is critical and is often the decisive factor in the applicability of computational fluid dynamics as a feasible analysis tool in design. Among the difficulties associated with increased problem complexity are the large number of degrees of freedom required to accurately predict the flow field.

One of the well practiced strategies to minimize the required mesh size for a specified accuracy category, is grid adaptation. The basic idea is to locally refine the grid in regions which most adversely affect the exactness of the final solution at the same time coarsening the grid in more unimportant regions to prevent incurring unnecessary computational costs. The usual practice is to adapt to certain physical features of the flow, such as shock waves, or stagnation points, by implementing pointers based on large flow gradients or some times second derivatives in combination with gradients. Yet, endless local refinement of the governing flow features does not necessarily insure that certain estimates of the global error will concurrently be reduced. Another strategy is to develop adaptive criteria based on *a posteriori* error estimates. Usual approaches has been to use local gradient recovery method to obtain higher order projections of gradients of the solutions. The solution is then analyzed with the higher order projection to estimate the error. For elliptic problems this procedure provides accurate error estimates using particular norms of the solution and its derivatives. For convection-dominated problems, as we encounter in aerodynamic applications, these method of error estimation yield inaccurate error approximation.

One of the alternative approach to make error estimation more relevant to engineering applications is to estimate the error made in predicting and integral quantity representing an output engineering interest. For example, in aerodynamic applications, this output might be the lift or drag. These approach invokes

*ESR, Department of Aerodynamics, Nowowiejska 24.

†Professor, Department of Aerodynamics, Nowowiejska 24.

the concept of duality, in which an equivalent adjoint formulation of the primal problem is utilized. The primary benefit of exploiting the dual problem, in the context of error estimation, is that the error in a selected functional can be directly associated to local residual errors of the primal solution through the adjoint variables. The error can be evaluated as an inner product of the local residual errors and the adjoint solution. This property unveils the potential to formulate optimal grid adaptive strategies designed to produce specially attuned grids for maximizing the accuracy of a specific cost function.

In principle, adjoints are computed via two methods: Continuous and Discrete. In the continuous method, adjoints are computed by deriving the adjoint equations of corresponding state variables and boundary conditions and then are solved using similar schemes as that of the primal. The discrete approach using Algorithmic Differentiation (AD)[3] is implemented either by source transformation or by an operator overloading tool. For an object-oriented code in C++ like RED, the use of an operator overloading tool becomes imperative. A convenience associated with the discrete approach is that the adjoint equation and corresponding boundary conditions need not be derived and discretized explicitly. Linear sensitivities of the cost function and the Jacobian matrix related with the primal residual are needed for the implementation of the procedure. The adjoint boundary conditions are automatically incorporated into the formulation via the primal residual operator. Experience from the different optimization procedure shows that adjoints of Residual Distribution Scheme[1] like algorithms have high computational cost even after employing standard performance improvement techniques. Intermediate values from the whole iteration history need to be stored in order to obtain the sensitivities. Thus, In this paper we also look at effective techniques[4] to tackle this problem. Symbolic treatment of the non-linear solver further reduces memory requirement largely with minor increment in runtime. As well as it provides adjoint vector which is key to the Output-Based Error Estimation.[2]

References

[1]Rémi Abgrall. Residual distribution schemes: current status and future trends. *Computers & Fluids*, 35(7):641–669, 2006.

[2]Krzysztof J. Fidkowski. Output-based error estimation and mesh adaptation for steady and unsteady flow problems. *VKI Lecture Series*, 41(4):122, 2015.

[3]Andreas Griewank and Andrea Walther. *Evaluating derivatives: principles and techniques of algorithmic differentiation*. Siam, 2008.

[4]Uwe Naumann, Johannes Lotz, Klaus Leppkes, and Markus Towara. Algorithmic differentiation of numerical methods: Tangent and adjoint solvers for parameterized systems of nonlinear equations. *ACM Transactions on Mathematical Software (TOMS)*, 41(4):26, 2015.

Advances in CFD Discretisation Schemes and Solution Algorithms for a Stable Discrete Adjoint

Mattia Oriani* and Guillaume Pierrot†

ESI Group, 90 Rue des Solets, Rungis 94513, France

Researchers in the field of CFD adjoint-based optimization are currently facing a number of challenges,[6] regardless of the specific approach. Examples of such difficulties, affecting specifically the adjoint Navier-Stokes equations, are: continuous adjoint solvers suffering from convergence issues, suggesting, among other causes, that the discretisation schemes and algorithms employed in standard CFD are not always suited to the nature of adjoint equations; discrete adjoint solvers also failing to converge in large cases unless a very well converged primal flow field is provided; adjoint codes produced via reverse Automatic Differentiation (AD) being based on hypotheses often not valid, thus producing unreliable gradient values as well as being costly in terms of storage memory. Evidence suggests that convergence issues in adjoint solvers are related to convergence of the CFD itself, meaning that classical Finite Volume (FV) schemes yield a solution that, although acceptable as a mere aerodynamics case study, cannot be used to produce a robust adjoint system. It appears therefore that adjoint solvers could benefit from a well-converged primal flow solution, which in turns requires a) an appropriate, mathematically sound discretisation scheme and b) an adequate solving algorithm. We attempted to tackle both of these aspects, and we summarize our main findings in the present work.

I. Discretisation schemes

In recent years a new family of CFD discretisation schemes, the Virtual Elements Method[1] (VEM) has emerged as a promising alternative to classical schemes such as FV. The aim is to allow for more freedom in the numerical model, most notably in mesh geometry (e.g. the possibility to use strongly non-orthogonal or non-convex elements) and discontinuity of material properties. The method also eliminates certain numerical artefacts typical of FV (e.g. Non-Orthogonal Correctors), thus ensuring robustness and improving convergence properties, without at the same time having to resort to certain somewhat constraining features of classical Finite Elements (FE), such as shape functions. The method is of particular relevance in the context of CFD optimization, as a) it can deal with mesh distortion as typically encountered in shape optimization processes, and b) it might alleviate robustness issues linked with the (continuous or discrete) adjoint Navier-Stokes equations, often used to compute gradients in gradient-based optimization. VEM has been largely and successfully validated for pure anisotropic diffusion operators,[2] convection-diffusion problems and 1st-order Navier-Stokes.[3]

We outline here our own incompressible Navier-Stokes scheme, a VEM-based method we named Mixed Hybrid Finite Volumes (MHFV). Compared to previous literature, our scheme a) features an original approach to derive and stabilize the VEM diffusion operator, b) is extended to 2nd-order accuracy for both velocity and pressure fields, and c) offers a selection of stabilization schemes for convective terms inspired by traditional FV and FE strategies. Results are presented on a series of 2D and 3D benchmark test cases, highlighting in particular how the scheme is reliable even on highly distorted meshes - a further appealing feature when it comes to shape optimization, where FV-compatible mesh quality cannot always be ensured by standard mesh-morphing tools.

II. Solution algorithms

The efficiency of traditional SIMPLE-like preconditioners is debatable: they are somewhat stable but they exhibit a rather poor convergence rate, they are prone to stagnation and they are affected by mesh

*mor@esi-group.com

†gpi@esi-group.com

refinement. They mostly owe their popularity to legacy reasons, being amongst the first devised working methods, and to their segregated nature, as they require solving linear systems that are relatively small and easy to deal with compared to the full Oseen-type system typically stemming from the Navier-Stokes discretisation.

Recent progress in computational power and linear solver capabilities led researchers to reconsider some other Navier-Stokes solution schemes, previously investigated but deemed unfeasible in industrial contexts. Research has successfully produced a number of alternative algorithms, although mostly restricted so far to the FE community, such as those based on the so-called approximate commutators.[4] Several interesting comparisons amongst various Navier-Stokes preconditioners have also been published.[8]

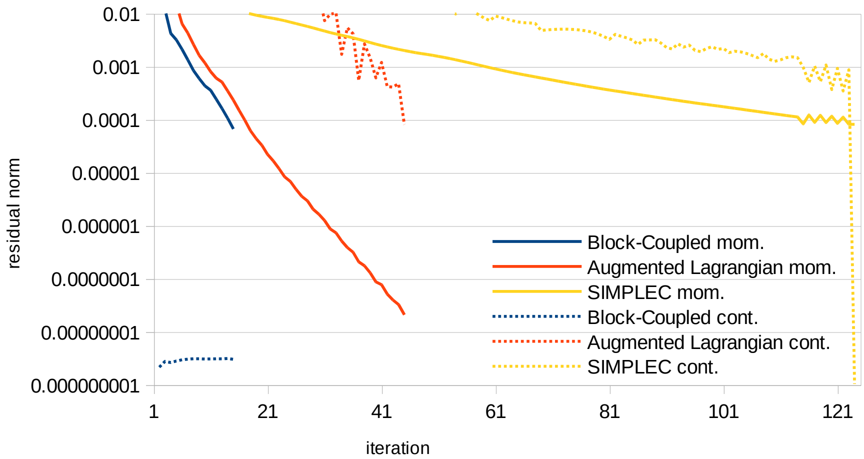


Figure 1. S-Bend 3D test case: convergence history of the primal x-momentum and continuity residuals for different solution algorithms.

We present here our own implementation of a traditional SIMPLE-like scheme as well as some alternative ones, including the Augmented Lagrangian[5] (AL), applied to the above mentioned MHFV Navier-Stokes solver. Performance is assessed via comparisons on a series of test cases (Figure 1).

We also investigate efficient ways of solving the discrete adjoint Navier-Stokes problem itself, which is in essence a linear system similar to the original Oseen, and is therefore affected by the same practical difficulties. We explicitly assemble the adjoint system via our Equational Differentiation[7] (ED) approach, and we devise multiple solution strategies (SIMPLEC, V-Coupled, AL) adapted from the primal. We run a series of adjoint test cases and compare performance of various solution schemes.

References

[1] Brezzi, F., Falk, R.S., and Marini L.D., *Basic Principles of Mixed Virtual Element Method*, Mathematical Modeling and Numerical Analysis, Vol. 48, No. 24, 2014, pp. 1227-1240.

[2] Brezzi, F., Lipnikov, M., and Shashkov M., *Convergence of the Mimetic Finite Difference Method for Diffusion Problems on Polyhedral Meshes*, SIAM Journal on Numerical Analysis, Vol. 43, No. 5, 2005, pp. 1872-1896.

[3] Droniou, J., and Eymard, R., *Study of the Mixed Finite Volume Method for Stokes and Navier-Stokes Equations*, Numerical Methods for Partial Differential Equations, Vol. 25, No. 1, 2008, pp. 137-171.

[4] Elman, H., Howle, V.E., Shadid, J., Shuttleworth, R., and Tuminaro, R., *Block Preconditioners Based on Approximate Commutators*, SIAM Journal of Scientific Computing, Vol. 27, No. 5, 2006, pp. 1651-1668.

[5] Olshanskii, M.A., and Benzi, M., *An Augmented Lagrangian Approach to Linearized Problems in Hydrodynamic Stability*, SIAM Journal on Scientific Computing, Vol. 30, 2008, pp. 1459-1473.

[6] Othmer, C., *Adjoint Methods for Car Aerodynamics*, Journal of Mathematics in Industry, Vol. 4, No. 6, 2014, pp. 1-23.

[7] Pierrot, G., *Equational Differentiation of Incompressible Flow Solvers as a Middle Ground between Continuous and Discrete Adjoint Methodologies*, WCCM XIECCM VECFD VIIACM-ECCOMAS, Barcelona (Spain), July 20-25, 2014.

[8] Segal, A., Rehman, M., and Vuik, C., *Preconditioners for Incompressible Navier-Stokes*, Numerical Mathematics: Theory, Methods and Applications, Vol. 3, No. 3, 2010, pp. 245-275.

A comparison of node-based and CAD-based parametrisations in shape optimisation

Rejish Jesudasan and Mateusz Gugala and Orest Mykhaskiv and Jens-Dominik Müller
Queen Mary University of London, SEMS, E14NS, UK

Salvatore Auriemma and Herve Legrand
OpenCASCADE, Guyancourt, 78280, France

Mladen Banovic and Andrea Walther
University of Paderborn, Paderborn, 33098, Germany

We present two parametrisations: one can be derived directly from the CFD mesh and another one based on parametric CAD-model. Shape optimisation is performed with both approaches coupled to a discrete adjoint solver and results are compared.

I. Introduction

Aerodynamic shape optimisation with gradient-based methods is rapidly gaining popularity in the aerospace and automotive industries. The most effective method to compute the required sensitivities is the adjoint method which allows to compute sensitivities of an arbitrary number of design variables at constant computational cost. This in turn opens up a wide range of possibilities to parametrise the shape as the number of design variables is no longer a limiting factor as long as the parametrisation algorithm can also differentiated in reverse mode.

A wide range of parametrisations have been proposed for aerodynamic shape optimisation.[1] On the one hand we will consider a node-based parametrisation which uses the surface nodes of the CFD grid as design variables. On the other hand we will consider CAD-based parametrisations based on open-source CAD-kernel OpenCascade Technology (OCCT).

II. Geometric shape parametrisations

Most shape parametrisation methods require manual setup. Setting up auxiliary grids for lattice-based methods, such as e.g. auxiliary grids with Hicks-Henne bumps on aerofoils or stacked spline curves for turbomachinery blades, involve substantial effort and are difficult to extend to complex geometries. Free-form deformations such as volume splines require the definition of auxiliary hexahedral volume grids that need to be snapped to the geometry to preserve features.

Adjoint methods do not penalise the size of the design space, hence we can consider very large spaces that guarantee to incorporate the largest possible number of degrees of freedom. With limitations on the design space being alleviated we can consider fully automatic parametrisations that do not require any manual definition by the user but can be derived from existing data.

In the node-based parametrisation, displacements of the surface grid nodes are the design variables which offers the richest design space the CFD discretisation can consider. As a matter of fact, this design space is even too rich for the CFD as the parametrisation method can express high-frequency modes which are not adequately resolved by the CFD and hence remain poorly damped. Additional regularisation is necessary, and implicit[2] as well as explicit[3] smoothing methods have been proposed, both of them requiring to tune a smoothing coefficient. The disadvantage of the method is that the optimal shape exists only as a mesh, transcription to CAD is not straightforward and any approximation will incur a loss of optimality.

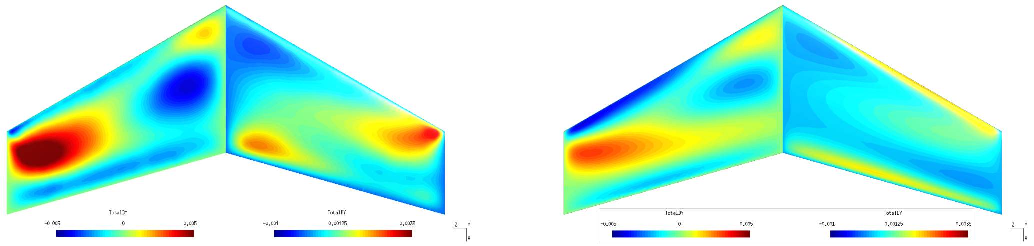
As the alternative, the CAD based approach NSPCC[4] works with the CAD geometry in the optimisation loop and produces the optimal shape in CAD. NSPCC considers movements of the control points of the NURBS patches of the BRep to alter the shape. The resulting design space is hence the richest space the provided BRep can express.

Furthermore, the optimisation of a parametric CAD-model is possible if the corresponding derivative information is provided. While this is usually not the case for commercial CAD products, we have used a differentiated version of OpenCascade Technology.[5] Access to OCCT source code enabled the use of automatic differentiation software ADOL-C, which facilitated gradient calculations. Hence, geometries parametrised in this CAD tool are equiped with sensitivities needed for optimisation.

III. Results

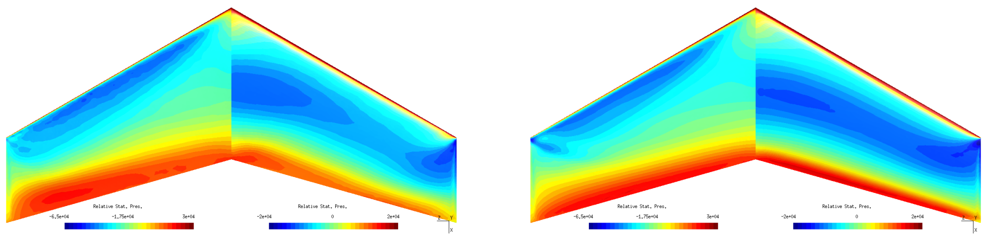
In this paper performance, efficiency and capabilities of the both parametric CAD and node-based methods are discussed and compared on a U-Bend testcase using an in-house flow and adjoint solver for the compressible Navier-Stokes equations.

Comparison of the node-based and CAD-based NSPCC approach for the inviscid transonic flow over an Onera M6 wing are shown in Figs. 1 and 2. The objective is to minimise drag subject to constant lift. The node-based method uses the displacement of 26,000 surface nodes regularised with 20 sweeps of explicit smoothing,[3] The CAD-based method uses two patches joining at leading and trailing edge with 13×12 points each, in total 2×468 DoF.



(a) *Node-based* (b) *CAD-based*

Figure 1: Drag minimisation of M6 wing, shape displacements for node-based (left) and CAD-based parametrisation (right).



(a) *Node-based* (b) *CAD-based*

Figure 2: Drag minimisation of M6 aerofoil, pressure for node-based (left) and CAD-based parametrisation (right).

Although the design spaces have vastly different sizes, the comparison for the top surface shows that very similar displacement modes are found by both parametrisation methods. The displacements of the lower surface exhibit differences, but the flowfield demonstrates that the objective function in both cases is very similar, it has low sensitivity against the bottom shape and is most likely multi-modal.

The presentation will focus on the comparison of parametric CAD and node-based representation of duct cases.

References

[1] Samareh, J., "Survey of Shape Parameterization Techniques for High-Fidelity Multidisciplinary Shape Optimization," *AIAA Journal*, Vol. 39, No. 5, 2001, pp. 877–884.

[2] Jameson, A. and Vassberg, A., "Studies of alternative numerical optimization methods applied to the brachistochrone problem," *Computational Fluid Dynamics Journal*, Vol. 9, No. 3, 2000.

[3] Jaworski, A. and Müller, J.-D., "Toward modular multigrid design optimisation," *Lecture Notes in Computational Science and Engineering*, edited by C. Bischof and J. Utke, Vol. 64, Springer, "New York, NY, USA", 2008, pp. 281–291.

[4] Xu, S., Jahn, W., and Müller, J.-D., "CAD-based shape optimisation with CFD using a discrete adjoint," *International Journal for Numerical Methods in Fluids*, Vol. 74, No. 3, 2013, pp. 153–68.

[5] Mykhaskiv, O., Müller, J.-D., Auriemma, S., Legrand, H., Banovic, M., and Walther, A., "Shape Optimisation with differentiated CAD-kernel for U-bend testcase," *NOED*, 2016.

Parametric CAD model based shape optimization using adjoint functions

Dheeraj Agarwal[1], Christos Kapellos[2], Trevor Robinson[1], Cecil Armstrong[1]

[1]School of Aerospace and Mechanical Engineering, Queen's University of Belfast, UK

[2]Volkswagen AG, Group Research, CAE Methods, Wolfsburg Germany

Abstract

Adjoint methods have proven to be an efficient way of calculating the gradient of an objective function with respect to a shape parameter for optimisation, with a computational cost nearly independent of the number of the design variables [1]. The approach in this paper links the adjoint surface sensitivities (gradient of objective function with respect to the surface movement) with the parametric design velocities (movement of the surface due to a CAD parameter perturbation) in order to compute the gradient of the objective function with respect to CAD variables.

For a successful implementation of shape optimization strategies in practical industrial cases, the choice of design variables or parameterisation scheme used for the model to be optimized plays a vital role. Where the goal is to base the optimization on a CAD model the choices are to use a NURBS geometry generated from CAD modelling software, where the position of the NURBS control points are the optimisation variables [2] or to use the feature based CAD model with all of the construction history to preserve the design intent [3]. The main advantage of using the feature based model is that the optimized model produced can be directly used for the downstream applications including manufacturing and process planning.

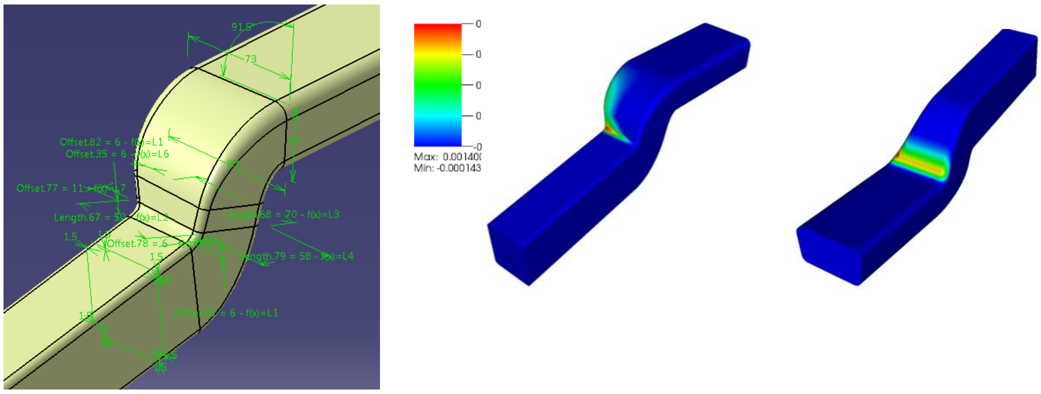
This paper presents an approach for optimization based on the feature based CAD model, which uses CAD parameters defining the features in the model geometry as the design variables. In order to capture the CAD surface movement with respect to the change in design variable, the "Parametric Design Velocity" is calculated, which is defined as the movement of the CAD model boundary in the normal direction due to a change in the parameter value.

The approach presented here for calculating the design velocities represents an advancement in terms of capability and robustness of that described by Robinson et al. [3]. The process can be easily integrated to most industrial optimisation workflows and is immune to the topology and labelling issues highlighted by other CAD based optimisation processes. It considers every continuous ("real value") parameter type as an optimisation variable, and it can be adapted to work with any CAD modelling software, as long as it has an API which provides access to the values of the parameters which control the model shape and allows the model geometry to be exported. To calculate the movement of the boundary the methodology employs finite differences on the shape of the 3D CAD models before and after the parameter perturbation. The implementation procedure includes calculating the geometrical movement along a normal direction between two discrete representations of the original and perturbed geometry respectively. Parametric design velocities can then be directly linked with adjoint surface sensitivities to extract the gradients to use in a gradient-based optimization algorithm.

The optimisation of a flow optimisation problem is presented, in which the power dissipation of the flow in an automotive air duct is to be reduced by changing the parameters of the CAD geometry created in CATIA V5, Fig. 1a. The flow sensitivities are computed with the continuous adjoint method for a laminar and turbulent flow [4] and are combined with the parametric design velocities (Fig. 1b) to compute the cost function gradients. A line-search algorithm is then used to update the design variables and proceed further with optimisation process.

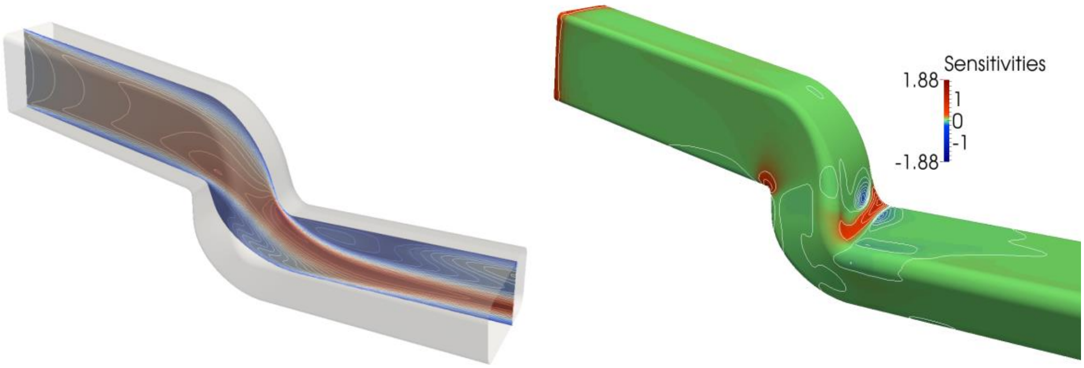
Keywords: CAD, design velocity, shape optimization, continuous adjoint

Corresponding Author: [1]d.agarwal@qub.ac.uk



(a) (b)

Fig 1. (a) Automotive air duct Model in CATIA V5 (b) Parametric design velocity field



(a) (b)

Fig 2. (a) Automotive air duct geometry and velocity profile (b) Surface sensitivities calculated with the continuous adjoint method. To reduce power dissipation, areas with positive values (red) have to be pulled out and areas with negative values (blue) to be pushed inwards

Preliminary Results

Optimization results with initial developments (using steepest descent) have shown good results resulting in a reduction of objective function (power dissipation) by nearly 4 % for laminar flow simulations. Further reduction is expected on using a suitable line-search algorithm in the optimization framework.

References:

1. Carsten Othmer, Adjoint methods for car aerodynamics. Journal of Mathematics in Industry, (2014) 4:6
2. Xu, S., Jahn, W. and Müller, J.-D. (2014), CAD-based shape optimisation with CFD using a discrete adjoint. Int. J. Numer. Meth. Fluids, 74: 153–168
3. Robinson, Trevor T., et al. "Optimizing parameterized CAD geometries using sensitivities based on adjoint functions." Computer-Aided Design and Application 9.3 (2012): 253-268
4. Georgios K. Karpouzas, Evangelos M. Papoutsis-Kiachagias, Thomas Schumacher, Eugene de Villiers, Kyriakos C. Giannakoglou, Carsten Othmer: Adjoint Optimization for Vehicle External Aerodynamics, *International Journal of Automotive Engineering*, Vol.7,No.1,pp.1-7(2016)

A hybrid adjoint shape sensitivity analysis of fluid-structure interaction problems

Reza Najian Asl* and Kai-Uwe Bletzinger-†
Chair of Structural Analysis, Technical University of Munich, Arcisstr. 21, 80333Munich, Germany

Large scale shape optimization involving the solution of an adjoint system of equations is an active area of research in computational fluid and structural dynamics, particularly for aerospace applications. In this regard, it is vital to have at hand a sensitivity analysis approach which shows stability, efficiency and accuracy. This contribution addresses a hybrid adjoint shape sensitivity analysis for fluid-structure interaction (FSI) problems which benefits from the continuous adjoint method for the fluid and the discrete adjoint method for the structure. The incompressible fluid flow is modeled using the Reynolds-averaged Navier-Stokes equations. For the structure, large displacements are considered due to the interaction with the fluid domain, resulting in geometrically nonlinear structural behavior and nonlinear interface coupling conditions. In the proposed coupled sensitivity analysis, two core ideas are introduced: 1) using a strictly continuous adjoint approach to derive adjoint boundary conditions for the FSI boundary, 2) formulating a force-type objective function over the far-field boundary which does not undergo any shape change.

Our formulation eliminates the need of expensive computation of partial derivatives such as the coupling terms in the coupled adjoint analysis. Instead it requires only to exchange boundary information like in the primal FSI solution and hence easily integrates into a partitioned co-simulation environment. Furthermore, the boundary conditions for a continuous adjoint fluid simulation using a far-field force objective have been derived and compared to a regular near field approach. It has been shown that both yield the same gradient information. Last but not least, the accuracy of the proposed coupled sensitivity analysis is verified through comparison of results with those obtained using a finite difference technique.

Communicating Author: reza.najian-asl@tum.de
*Ph.D. Candidate, Chair of Structural Analysis, Technical University of Munich.
† Professor, Head of Chair of Structural Analysis, Technical University of Munich

Surrogate-Model Assisted Evolutionary Optimization of an Axial Compressor Stator

Mohamed hassanine Aissa* Christopher Chahine†

Von Karman Institute for Fluid Dynamics, Rhode-Saint-Genese, 1640, Belgium

Tom Verstraete‡

Queen Mary University of London, London, E1 4NS, United Kingdom

In this paper we present the aerodynamic shape optimization of the TurboLab Stator test case proposed for the present conference. A metamodel assisted Differential Evolution algorithm is applied coupled to a GPU-accelerated Computational Fluid Dynamics code for the aerodynamic performance evaluation of the annular stator geometry. The stator is parameterized utilizing common turbomachinery blade parameters such as chord length, chordwise blade metal angle distributions and thickness distributions on several spanwise blade sections. Lean and sweep are applied as three dimensional design features. A loss reduction was obtained with the optimized geometry along with an improvement of the axial outflow.

I. Introduction

The optimization system used in present work is the result of more than one and a half decades of research and development at the von Karman Institute[1] . Its core components are a multi-objective Differential Evolution algorithm[2] , a database, several metamodels and a high-fidelity evaluation chain. The evaluation chain comprises a fully automatic geometry and CAD generation, an automatic meshing and a high-fidelity performance evaluations by Computational Fluid Dynamics (CFD). We present a new feature for the optimization system consisting concerning the acceleration of the CFD evaluation. For this purpose we deploy Graphical Processing Unis (GPUs), known for their high computational power, to accelerate an in-house RANS solver with implicit time stepping. The GPU-accelerated solver as part of the established solver has been used to optimize the TurboLab Stator test case.

II. Metamodel assisted Optimization

The TurboLab is a low Mach number compressor stator from Technical University of Berlin. Its main task is to turn the inlet swirling flow into an axial outflow with a minimal total pressure loss. The stator geometry has been parameterized by a set of parametric Bézier curves. Two curves describe the thickness and the chord length distributions at 0 and 100% span. The camberline based on the metal angle distribution of the blade is defined at 0, 50 and 100% span giving the inlet and outlet angle in addition to the angle at 2% of the axial chord. Lean and sweep are defined as 3D effects and a set of Bézier curves describes the casing endwall and the hub. While the casing endwall has a constant radius, the hub presents a contouring defined by a Bézier spline with 4 control points: at the inlet, leading edge, trailing edge and at the outlet. The chord length is kept constant as required in the testcase and the thickness distribution is not changing spanwise. 21 design variables (DV) have been selected for the optimization namely the metal angle distribution of the camberline accounting for 9 DV, the thickness distribution accounting for 2 DV, lean and sweep contributing with 6 extra DV and finally a hub contouring accounting for 4 DV.

The CFD evaluation has been carried out by two solvers first FINE$^{\mathrm{TM}}$ a commercial RANS solver from Numeca on CPU with explicit time stepping and second an in-house CFD solver with GPU acceleration and implicit time stepping. The two CFD solvers are not cooperating during the optimization and their

*PhD Candidate, Turbomachinery and Propulsion Department, Waterloosesteenweg 72, B-1640 Sint-Genesius-Rode
†DPhil Candidate, Turbomachinery and Propulsion Department, Waterloosesteenweg 72, 1640 Sint-Genesius-Rode, Belgium and Department of Engineering Science, University of Oxford, Osney Mead, Oxford, OX2 0ES, United Kingdom.
‡Visiting Professor, School of Engineering and Material Science, E1 4NS Mile End Road, London, United Kingdom.

redundant use is related to a benchmark of the GPU technology in design optimization, which is part of this work. The GPU accelerated steady RANS solver follows a spacial discretization based on finite volume approach. Convective fluxes are calculated using Roe upwind approximation while second order accuracy is achieved through the MUSCL approach. Viscous fluxes are approximated following a central discretization scheme and the source term contains contribution from Spalart-Allmaras (SA) one-equation turbulence model. The time integration is performed using an 3 stages implicit Runge-Kutta first order scheme. The commercial solver has a similar space discretization with a low Mach number preconditioning of the right hand side. The time integration differs also since FINE$^{\text{TM}}$ uses the explicit time stepping multistage Runge-Kutta scheme. For both solvers total quantities are imposed at the inlet along with the flow direction and the mass flow is imposed at the outlet. Every evaluation provides a mass flow averaged total and static pressures at inlet and outlet. These are used to compute the total pressure loss defined as $Loss = (p_{01} - p_{02})/(p_{01} - p_1)$. The area averaged axial deviation is also provided.

The first objective of the optimization is to reduce the total pressure loss and the second objective is to reduce the deviation of the outflow as the integral of the whirl angle squared. The optimization takes into account 3 operating points with different inlet whirl angles: -47, -42 and -37 degree. The first operating point with the nominal whirl angle has a weight of 0.5 for the optimization objectives while both other operating points have a weight of 0.25 each. The constraints are of two types: CFD and manufacturing constraints. The CFD constraint concerns the mass flow of the full annulus, which has to be 9 kg/s with a tolerance of 0.1 kg/s. We impose the prescribed mass flow as an outlet boundary condition which ensures that every converged CFD meets this requirement. For the manufacturing constraints a geometry check has been implemented and integrated in the automatic tool chain. These constraints fix the number of blades, the axial chord length and impose some requirements for the blade thickness. The blade should be thick enough to have a place for 2 cylinders of material for the fixture on hub and shroud with a radius of 5 mm and 20 mm of depth. The distance between the two holes is fixed to 60 mm. An other constraint concerns the hub contouring for which the radius change is limited by -5 mm and +10 mm. This limit has been introduced in the hub parameterization restricting the vertical translation of the control points.

III. Results

Figure 1 shows a plot of the objective space after 7 iterations (198 CFD evaluations). All plotted designs satisfied both the aerodynamic and the manufacturing constraints. Different points from the Pareto front dominates the baseline designs, which is nevertheless already highly optimal since it is very close to the Pareto front. As usual a trade-off has to be met for the selection of a design from the Pareto front but the design that is non-dominated by any other design achieved an improvements of 0.07% in total pressure loss and 6% in outflow angle.

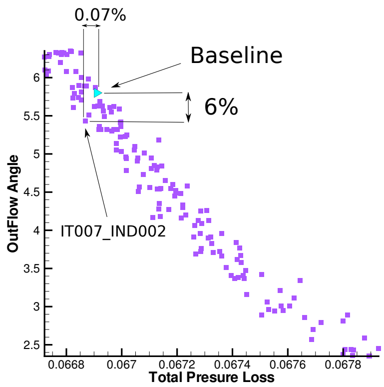


Figure 1. Plot of the objective space showing the baseline design compared to a non-dominated design and other designs generated during the optimization

References

[1]T. Verstraete, Multidisciplinary Turbomachinery Component Optimization Considering Performance, Stress and internal Head transfer, PhD thesis, University of Ghent/ Von Karman Institute for Fluid Dynamics, 2008.

[2]Storn R, Price K. Differential evolutiona simple and efficient heuristic for global optimization over continuous spaces. Journal of global optimization. 1997 Dec 1;11(4):341-59.

Adjoint-Based Optimization of Multi-Stage Turbomachines using SU2

Tim Albring*, Nicolas R. Gauger†
Chair for Scientific Computing, University of Kaiserslautern, Germany

Salvatore Vitale‡, Matteo Pini§
Propulsion and Power, Delft University of Technology, Netherlands

The increasing computing power during the last decades enabled the detailed analysis of complex physical problems using computational fluid dynamics (CFD). It is especially valuable for the turbomachinery design process. However, it also becomes considerably harder to further improve the efficiency by solely relying on the aerodynamicist's experience and knowledge. At this stage gradient-based optimization methods apply as they have the potential to aid the designers in the development of highly-efficient compressors and turbines. This contribution presents the current state of the extension of the open-source multi-physics framework SU2 to treat the analysis and optimization of multi-stage turbomachines. In particular we rely on the Algorithmic Differentiation features to develop an adjoint solver that is capable of automatically linearizing the flow solver including the mixing-plane formulations, complex thermo-physical laws and turbulence models. Hence we completely avoid the error-prone hand linearization.

I. Introduction

Modern CFD codes for the analysis of flows in turbomachines are typically quite complex, as they have to include features like rotating frames, non-reflecting boundary conditions, periodic boundaries and many more. The linearization of these analysis tools, be it on the continuous or discrete level, is even more challenging and prone to errors. Recently Walther and Nadarajah[1] showed the derivation and application of a discrete adjoint method using an explicit lineariziation of the mixed-out flow variables, the characteristic boundary conditions and the corresponding boundary fluxes at the mixing-plane. In contrast to the present work, they were using the frozen turbulent viscosity assumption and validated the gradients only for the inviscid case. Here we present the extension of the open-source framework SU2 for the analysis and optimization of turbomachines using an exact linearization of all flow solver features.

II. Optimization of Turbomachines with SU2

SU2 is a suite of open-source software tools for the numerical solution of partial differential equations (PDE) and performing PDE constrained optimization. Initially developed at Stanford University it now exhibits collaborations from all over the world. Native support for Algorithmic Differentiation (AD) was recently added (Albring et al.[3]) as well as the possibility to simulate turbomachines (Vitale et al.[2]). The discrete adjoint solver is based on the fixed-point formulation of the underlying flow solver which results in a method commonly denoted as *Duality-Preserving Iteration* or *Reverse Accumulation*:

$$\bar{U}^{n+1} = \frac{\partial}{\partial U} J^T(U^*, X) + \frac{\partial}{\partial U} G^T(U^*, X)\bar{U}^n, \tag{1}$$

where G is the iterator of the flow solver and J is the objective function. U^* represents the (numerical) solution of the RANS and turbulent equations and X are the mesh node coordinates. All occurring derivatives are computed by applying AD on the top-level iteration, thus they include all features of the flow solver including the transfer of information between the zones at the mixing-plane interface. Using

*PhD student, tim.albring@scicomp.uni-kl.de
†Professor
‡PhD student, s.vitale@tudelft.nl
§Professor

local preaccumulation[4] and expression templates[5] the resulting solver yields comparable performance with hand-linearized discrete adjoint codes.

III. Preliminary Results and Outlook

To get the first impression of the state of the development we applied the flow solver to a generic centrifugal stator-rotor combination in viscous turbulent flow modeled by the SST turbulence model. The two zones are coupled through a mixing-plane interface using mixed-out variables. Non-reflecting boundary conditions are applied at the inflow and outflow boundaries in each zone. Figure 1 shows the resulting relative Mach number contours as well as the mesh used for the computation. For the optimization we will use the Free-Form deformation approach to maintain a smooth variation of the surface of each blade. Exemplary definitions of these boxes are also shown in figure 1.

For this configuration we also computed the gradient on the surface of each blade with the total static efficiency of the stage as objective function. The resulting surface sensitivity is shown in figure 2.

In the final contribution we will show a validation of the solver and the corresponding gradients as well as the optimization of a real multi-stage configuration.

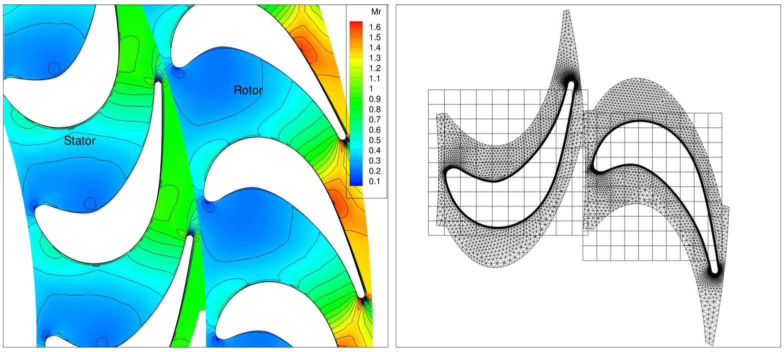


Figure 1. Relative Mach number contours (left) and computational mesh with exemplary FFD box definition (right).

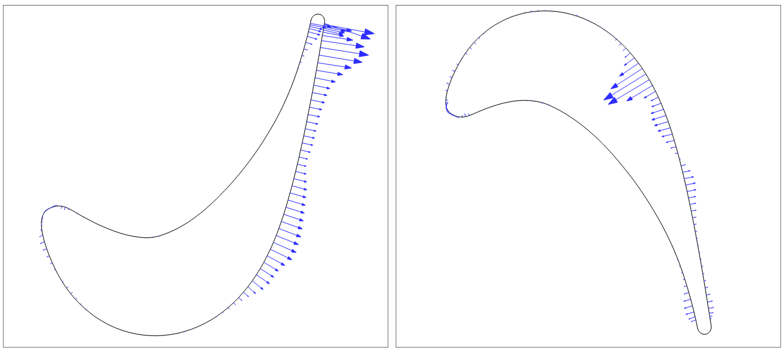


Figure 2. Relative surface sensitivity for the stator (left) and the rotor (right) with total static efficiency as objective function.

References

[1]Walther, B. and Nadarajah, S.. *Adjoint-Based Constrained Aerodynamic Shape Optimization for Multistage Turbomachines*, Journal of Propulsion and Power, Vol. 32, No.5, 2015.

[2]Vitale, S., Pini, M., Ghidoni, A., Colonna, P.. *Fluid Dynamic Design and Analysis of a Highly Loaded Centrifugal Rotor for Mini ORC Power Systems*, 3rd International Seminar on ORC Power Systems, Brussels, Belgium, 2015.

[3]Albring, T., Sagebaum, M., Gauger, N.R.. *Development of a Consistent Discrete Adjoint Solver in an Evolving Aerodynamic Design Framework*, AIAA Paper 2015-3240, 2015.

[4]Andreas Griewank and Andrea Walther. *Evaluating Derivatives: Principles and Techniques of Algorithmic Differentiation*, Other Titles in Applied Mathematics. SIAM, 978–0–898716–59–7., 2008.

[5]Albring, T., Sagebaum, M., Gauger, N.R.. *CoDi - Code Differentiation Package*, Retrieved from http://www.scicomp.uni-kl.de/software/codi (2016, April 15)

Overall Aircraft Design optimization of a new aircraft configuration

Andrea Viti*, Antoine Dumont† and Gerald Carrier‡
ONERA The French Aerospace Lab, 8 Rue des Vertugadins, 92190 Meudon, France

Thierry Druot§
Airbus France S.a.s, 316 Route de Bayonne, 3106 Toulouse, France

This paper aims at describing a Multidisciplinary approach for new aircraft configuration optimization. An overall aircraft optimization aiming at Direct Operative Costs (DOC) reduction will be performed on top of the Aero-structural optimization. Top level variables are chosen in such a way that the trade-off between aerodynamics and structure is found without considering strong coupling variables. For that reason, wing planform and thicknesses are optimized during an Overall Aircraft Preliminary Design phase.

Nomenclature

MDO	=	Multidisciplinary Design Optimization
DOC	=	Direct Operative Cost
OAD	=	Overall Aircraft Design

I. Introduction

The idea is to have the overall aircraft design analysis and optimization in top of the aero-elastic and structural optimization procedure so that the wing planform is optimized for cost reduction, and not for aero-structural needs only. The Overall Aircraft Design (OAD) optimization targets main wing variables that will be used as input for the further optimizations which will give back the sensitivity analysis of the aero-elastic and structural objective functions with respect to the OAD variables. At that point, cost reduction will be addressed again, and the loop repeated till cost convergence is reached.

II. Methodology

The *OAD – Aero-elastic – Structural* optimization is organized in such a way that each discipline is independent but each of them gets some results from the previous optimization level. Actually, the optimization workflow is as follow:

i. Multipoint aero-elastic and structural optimization of the FSW architecture and sensitivity analysis of the objective functions with respect to the OAD parameters;
ii. OAD optimization for decreasing the DOC to provide the new planform variables to be used for the further aero-elastic and structural optimization and sensitivity analysis;
iii. The optimized top level variables are fixed in the next aero-structural process that will provide new values and sensitivity analysis for the OAD optimization.

First of all, the structural optimization is performed in order to minimize the wing weight taking the stress level of each structural component under constraint. The wing load is recomputed over time in order to be consistent with the stiffness change. As shown in Figure 1, once the wing weight is optimized, the new aero-elastic equilibrium based on the new wing stiffness is reached, and the new wing load is used for the next structural optimization. The convergence is obtained when the difference between the optimized weight of the last optimization and the one before is less than 1%.

After that, a multipoint aero-elastic optimization has been set in order to increase the L/D ratio for 3 different cruise conditions. *Twist* and *camber* at different control sections are used as shape parameters. Viti et al [1] investigated on the most appropriate shape variables to be used for different flow conditions.

---

*PhD Candidate, DAAP/ACI, ONERA Meudon.
† Researcher, DAAP/ACI, ONERA Meudon.
‡ Director of the DAAP/ACI unit, ONERA Meudon.
§ Expert Engineer, Preliminary Design Office, Airbus Toulouse.

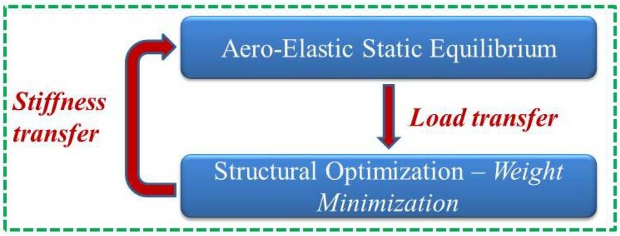


Figure 1: Structural Optimization at the Aero-elastic equilibrium

Concerning the cost reduction, the following 6 top level variables are chosen.

Wing surface	**S**
Aspect ratio	λ
Tape ratio	c_{root}/c_{tip}
Sweep angle	$\varphi_{25\%}$
Relative thickness at 40% of the span	$t/c_{\ 40\%\ span}$
Relative thickness at the tip	$t/c_{\ tip}$

Table 1: Top level variables for the OAD optimization

The aforementioned variables have direct impact on both aerodynamics and structure. Following the presented approach, the OAD optimization will chose the 6 top-level variables to address cost reduction and the following aero-structural process will increase the efficiency of the aircraft, enriching at the same time the physics of the top-level optimization. This iterative procedure will end once that the DOC of a previous iteration differs from the last by a certain error.

Multidisciplinary Optimization

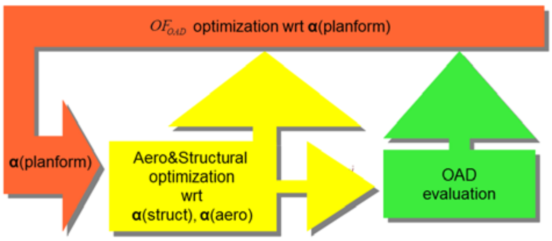


Figure 2: OAD top level optimization on top of the aero-elastic and structural ones; α(struct) and α(aero) are the structural and aerodynamic variables respectively

III. Results and Conclusions

In this paper a new MDO approach for preliminary aircraft design is presented. In particular, the top-level OAD process optimizes 6 wing variables for DOC reduction. Such variables are fixed during the following aero-structural optimization procedure. Preliminary results show the effectiveness of this approach to address simultaneous requirements of different disciplines without adding untreatable complexity and maintaining preliminary design characteristics such as fast and reliable analysis and optimization.

IV. Acknowledgements

This work was funded by the European Commission through the research project AMEDEO (Aerospace Multidisciplinarity Enabling Design Optimisation) under the FP7-PEOPLE Marie-Curie ITN 316394. The authors gratefully acknowledge all the financial and technical support provided by AMEDEO and the European Commission.

References

[1]A. Viti, A. Dumont, G. Carrier (2015). “Innovative Aero-Structural Design Procedure for a Forward-Swept Wing in Preliminary Design”, 50th 3AF International Conference on Applied Aerodynamics, Toulouse, France.

Node-based shape optimization in aircraft preliminary design

Daniel Baumgärtner* and Kai-Uwe Bletzinger†
Technical University of Munich, Arcisstr. 21, 80333 München, Germany

Andrea Viti‡ and Antoine Dumont§
ONERA The French Aerospace Lab, 8 Rue des Vertugadins, 92190 Meudon, France

We present an application of the Vertex Morphing Method to aircraft preliminary design. Vertex Morphing is a node-based shape optimization technique, that unifies shape control, mesh-regularization and sensitivity filtering. Since it directly uses surface nodes of the given discrete model, it is particularly useful to explore the design space at a minimum modelling effort. It also allows for effective shape optimization where another parametrization is not obvious or even infeasible. Hence, it represents an alternative to the application of an experience-based shape parametrization. Particular problem of interest is the aerodynamic shape optimization of a flexible wing. Given this multidisciplinary problem, we investigate the Vertex Morphing method in terms of possible design improvements and quality of the final design. Furthermore, we present a generic implementation of the Vertex Morphing method in an established open-source software framework dedicated to multiphysics in general. The implementation provides a standalone optimizer with a python interface, that allows any interested user to conveniently try Vertex Morphing for own problems from different physical applications (e.g. structures or aerodynamics). An outlook on the shape optimization of a forward-swept wing aircraft finally highlights capabilities of the implementation.

I. Introduction

The most direct approach to modify geometry in the context of shape optimization is to use the surface nodes of the discrete model as design handles. Such a node-based approach does not include any physically motivated parameterization but sets on high design freedom with possibly great optimization potential at no additional parameter assumptions. Typically, however, a node-based approach suffers from mesh dependency, mesh irregularity and non-smooth shape derivatives. To overcome this burden, Hojjat et. al suggested a method that unifies shape control, mesh regularization and sensitivity filtering in a single formulation referred to as the Vertex-Morphing Method [1]. Integral part of the Vertex Morphing method is the filtering. Different as in other node-based approaches, though, the effect of the filter by construction is purely local and instead of smoothing either the sensitivities or the shape updates, we map the quantities between design and geometry space, such that the optimization problem is not modified. Using this approach, we may explore different local minima by a single parameter, i.e. the filter size. As a matter of fact, Bletzinger showed in [2], that there is a perfect transition between the filtering approach and other CAD-based shape parameters. Using Vertex Morphing the full potential of node-based shape optimization may be exploited and significantly improved designs with possibly several alternatives may be found - even for large industrial-sized examples as e.g. discussed in [1], [3], [4].

II. Problem formulation

Problem of interest is the lift constrained drag minimization of a flexible ONERA M6 wing. We utilize the Euler equations for the primal fluid problem and the steady nonlinear structure equations for the primal structure problem. The present fluid-structure interaction is analyzed iteratively using the Gauss-Seidel method with Aitken relaxation. In terms of the sensitivity analysis, we follow a sequential single-disciplinary approach. That is, we compute in each optimization step the fully coupled problem, but neglect the corresponding effects on the sensitivities. In favor of computational efficiency and because we are given purely aerodynamic response

---

Communicating Author: daniel.baumgaertner@tum.de
* Ph.D. Candidate, Chair of Structural Analysis
† Professor and head of the Chair of Structural Analysis
‡ Ph.D. Candidate, Applied Aerodynamics Department
§ Research engineer, Applied Aerodynamics Department

functions, we rather compute the necessary surface sensitivities from the fluid problem only, using a continuous adjoint approach. As design variables we chose all surface nodes of the discrete wing model (~18,000). To run the corresponding optimization problem, we utilize a specifically modified projection algorithm. Particular benefit of this algorithm is, that it maintains the simplicity of unconstraint descent methods like steepest descent or CG (also w.r.t. its implementation), yet provides the ability to take into account constraints sufficiently accurate.

III. Software framework

We present "Kratos Shape", a standalone optimizer dedicated to shape optimization of arbitrary geometries using Vertex Morphing. The optimizer is implemented within the established open-source simulation framework Kratos [5]. Due to an existing python interface and a concept that strictly distinguishes between the involved physical analyses and the necessary geometry processing, the optimizer by construction is very flexible. It for example may be used with different simulation software, proprietary or open-source, or it may be applied to different physical disciplines, like for aerodynamic, structural or aero-structural shape optimization. Moreover, the optimizer is developed having in mind practical problems with millions of design variables. Particular focus is therefore on realizing a process that avoids unnecessary solver calls during the optimization. This is obtained by an implementation based on the interaction of three separated objects: an optimizer, a controller and an analyzer.

IV. Results

The results show that Vertex Morphing allows to identify optimization potential without any effort or assumption regarding a parametrization. For the given problem, we hence obtain a significantly improved wing design at satisfied constraint. The results also show that Vertex Morphing allows for an enhanced control of the wing shape. E.g. the wing thickness may be implicitly controlled by a specific choice of the filtering. Moreover, we are able to generate evidently C2-continuous surfaces. This is because the originally noisy sensitivities are translated into consistent, smooth shape updates, such that no surface irregularities arise (see Figure 1). Still the trailing edge remains sharp. As a general result we observe, that Vertex Morphing can be successfully applied in an aero-structural context, where the position of the surface nodes additionally depends on the solution of the involved coupled problem.

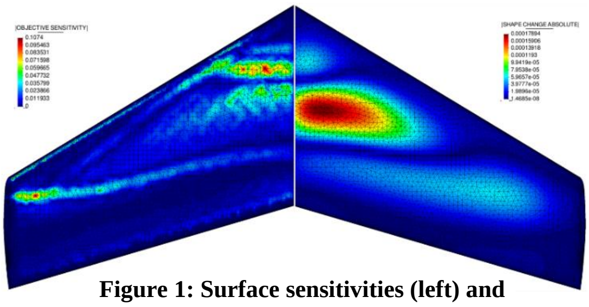


Figure 1: Surface sensitivities (left) and resulting shape updates (right)

Generally, it is evident from the results, that Kratos Shape enables node-based shape optimization in a multidisciplinary context. To highlight its flexibility, we additionally perform a constrained aerodynamic shape optimization of an entire forward-swept wing aircraft. Herein drag is reduced by ~20% while satisfying the constraint. The resulting shape is depicted in Figure 2. Note that for the aircraft we linked Kratos to the ONERA in-house CFD solver elsA, whereas for the M6 above, we linked to Kratos to the open-source CFD code SU2.

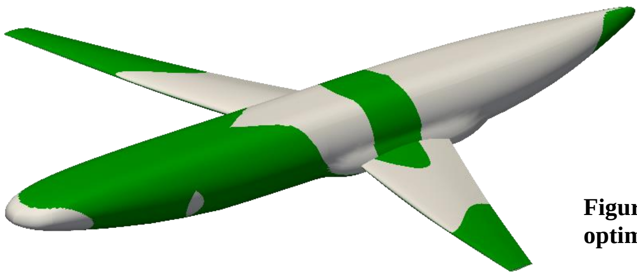


Figure 2: Superposed baseline (grey) and optimized shape (green)

V. References

[1] M. Hojjat, E. Stavropoulou, and K.-U. Bletzinger, "The Vertex Morphing method for node-based shape optimization," *Comput. Methods Appl. Mech. Eng.*, vol. 268, pp. 494–513, 2014.

[2] K.-U. Bletzinger, "A consistent frame for sensitivity filtering and the vertex assigned morphing of optimal shape," *Struct. Multidiscip. Optim.*, vol. 49, no. 6, pp. 873–895, 2014.

[3] M. Firl, "Optimal Shape Design of Shell Structures," *PhD Thesis*. Technische Universität München, 2010.

[4] D. Baumgärtner, A. Viti, A. Dumont, G. Carrier, and K.-U. Bletzinger, "Comparison and combination of experience-based parameterization with Vertex Morphing in aerodynamic shape optimization of a forward-swept wing aircraft," in *17th AIAA/ISSMO MAO*, 2016.

[5] International Center for Numerical Methods in Engineering, "Kratos Multi-Physics," 2016. [Online]. Available: http://www.cimne.com/kratos/.

Paper Title: A Study on the Effect of Several Modelling and Analysis Parameters on the Optimization of Composite Laminates for Vibro-acoustic Requirements

Paper Affiliation: ASMO-UK 2016

Presentation Type: Oral Presentation

Contact Information
Name: Gökhan Serhat
Organization: Koc University, Istanbul
Title: PhD Candidate
Postal address: Department of Mechanical Engineering, College of Engineering, Koc University, Istanbul, 34450, Turkey
Phone: +90 532-312-8965
e-mail: gserhat13@ku.edu.tr

Authors' Names and Affiliations: Gökhan Serhat, Tiago Goncalves Faria, Ipek Basdogan

Keywords: Laminated composites, simulation-driven optimization, finite element modelling, lamination parameters, optimization methods

Text of the Abstract

Optimization of laminated composite structures for mechanical and vibro-acoustic requirements often involves alteration of stiffness distribution and structure thickness. This process requires accurate models and methods. The results can change significantly depending on the values of the modelling parameters and the analysis methods used for the optimization. Hence having prior knowledge on their effect is essential for being well aware of the design methodology options and choosing the most appropriate one; also for validation and justification of the results. Therefore, the investigation and quantification of the modelling and analysis parameter effects in the optimization of composite structures is important.

The optimum design for the laminated composited can be found by analytical means for relatively simpler problems and by using finite element simulations for more complex ones. In order to work on a convex domain the modelling the technique called lamination parameters is used which provides means to represent the overall stiffness behavior. This removes the dependency of the optimum to the assumptions on the number of plies, ply thicknesses and initial configuration. Individual and combined effect of the parameters can be studied using design of experiment methods (DoE) methods.

In this paper, the several problem variables that affect the stiffness optimization results of composite panels are investigated. Design sensitivity analyses are conducted with respect to physical model parameters such as thickness and panel aspect ratio. In addition, the influence of the numerical solution parameters such as mesh density and initial design points is analyzed. The effects of each parameter on the several vibro-acoustic responses are quantified. In the optimization processes, DoE-based and gradient-based methods are used. Studies combining different responses and optimization methods have been carried out and their combined implications are presented.

A preliminary design method for optimizing composite forward-swept wings

Christopher Bach* and Reda Jebari†
Aeronautics Department, Imperial College London, SW7 2AZ. UK.
Andrea Viti‡
ONERA, 912190 Meudon, France and Aeronautics Department, Imperial College London.
Rob Hewson§
Aeronautics Department, Imperial College London, SW7 2AZ. UK.

In this work, we describe a preliminary design method for optimising composite forward-swept wings. The approach uses a simplified structural wing model based on the theory of thin walled anisotropic laminate cross-sections. This way, a cross-sectional stiffness matrix can be obtained which is used to couple the deformations with the aerodynamic forces. Bending-torsion coupling of the spar is used to avoid aeroelastic divergence, a typically encountered problem for forward swept wings. Optimisation was undertaken for different sweep angles, the weight of the wing spar being the objective function. A number of constraints were applied, including the avoidance of aero-elastic divergence.

I. Introduction

Aircraft wings have traditionally been swept backwards rather than forwards due to aeroelastic problems with the configuration, one of the most significant problems being divergence. However, if these problems can be addressed, Forward-Swept Wings (FSW) have a number of advantages, including better maneuverability and the potential to deliver a decrease in vortex drag due to weaker vortices at the wing tips.[5] The cabin layout can also be improved for smaller aircraft due to the more aft location of the wing root and therefore continuous spar, when compared to a rearward swept wing.

The work presented here demonstrates a fast means of obtaining an early estimate of possible weight savings by using composite tailoring for FSW aircraft.

II. Aeroelastic and structural models

The wing is divided along the span into a discrete number of sections, with the lift about the quarter chord obtained from the sections' lift curve slopes. The wing spar was modelled as a thin-walled cantilever beam with rectangular cross sections. It was assumed to carry all the loads. In order to obtain the relationship between beam's deformations and the applied aerodynamic loads, a model is developed from the theory of anisotropic thin-walled closed sections presented in.[6] The resulting model relates the cross-sectional forces and moments to the respective curvatures via a global stiffness matrix. This eventually leads to two coupled differential equations describing the twisting and bending of the wing due to the aerodynamic loads. The Ritz method has been used to predict the occurrence of aeroelastic divergence, as a closed-form solution to these equations can only be obtained for specific wing properties.[2]

III. Optimisation

Optimisation was undertaken on the composite stacking sequence, using a bi-level approach as presented by.[4] Lamination parameters (LP) are used as the first level design variables. A correspond-

*PhD Student
†former MSc Student in Composites
‡Marie Curie Early Stage Researcher and PhD Student
§Senior Lecturer

ing stacking sequence is matched to these parameters in a second step, using a permutation GA. The ply angles were restricted to the set of $\{0°, \pm 45°, 90°\}$, as is typical in many industrial applications. The minimum weight was sought, subject to the constraints below:

- No aeroelastic divergence
- No stalling: $\alpha < \alpha_{stall}$ at all wing span locations
- Twist angle $\bar{\theta}(\bar{y}) \leq 5°$ at all wing span locations
- Lift equal to lift required $\pm 1\%$
- Ply percentage in each pre-defined direction $\geq 10\%$
- No structural failure as estimated by the Tsai-Hill criterion (first-ply-failure, FPF)
- Feasible LP region as described by[3]

IV. Results

Typical results showed the variation in the stiffness along the length of the wing as shown in figure 1. It is interesting to note that the coupling coefficient, κ, leads to a nose down twist of the wing in response to aerodynamically applied bending moments. The reason for this coupling was the fraction of 45° plies in the top panel (and a corresponding number of $-45°$ plies in the bottom panel, as observed from the exterior of the wing-box). This is illustrated in figure 2. Results also showed the bi-level optimisation strategy to be more efficient than a computationally expensive approach of directly optimising the stacking sequence of the composite spar. Compared to an optimised design achieved with quasi-isotropic layups and hence no bend-twist coupling, a weight saving of 13% was found for the wing spar, at a forward sweep angle of -25° and at Mach 0.7.

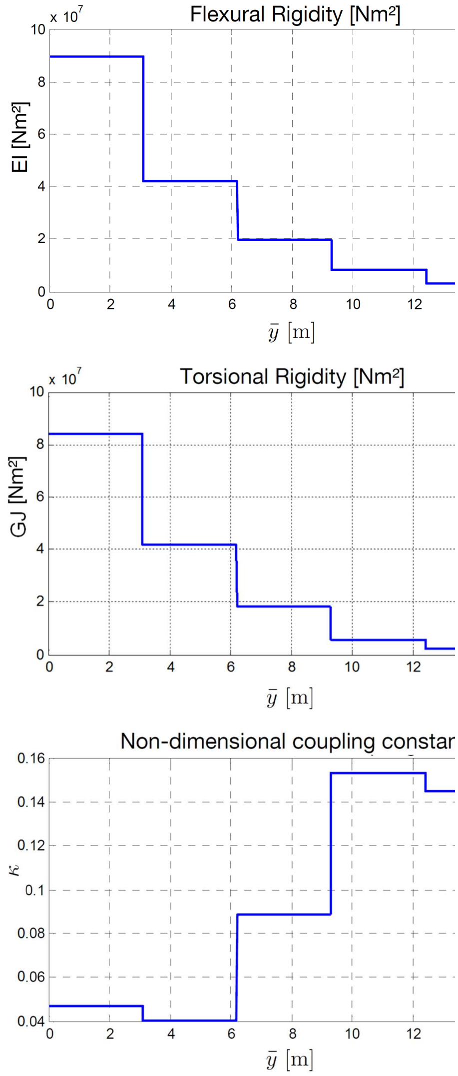


Figure 1: Optimum stiffnesses EI, G and κ. Results obtained for $\Lambda = -15°$, $M_\infty = 0.8$

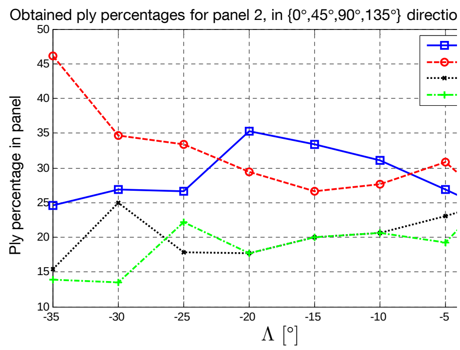


Figure 2: Ply percentages in top panel of cretization segment 2 vs. Λ, at $M_\infty = 0.8$.

References

[1]Bach, C. et al., *Composite stacking sequence optimization for aeroelastically tailored forward-swept wings*, Accepted Structural & Multidisciplinary Optimization, 2016.

[2]Bisplinghoff, R. L. et al., *Aeroelasticity*, Dover Publishing, 1955.

[3]Diaconu, C.G. and Sekine, H., *Layup Optimization for Buckling of Laminated Composite Shells with Restricted Layer Angles*, AIAA Journal, **10**:42, 2004.

[4]Liu, D. et al., *Bilevel Optimization of Blended Composite Wing Panels*, Journal of Aircraft, **48**:1, 107-118, 2011.

[5]Johnsen, F. A., *Sweeping Forward: Developing & Flight Testing the Grumman X-29A Forward Swept Wing Research Aircraft*, NASA Aeronautics Book Series, 2013.

[6]Salim, H. and Davalos, J. *Torsion of open and closed thin-walled laminated composite sections*, Journal of Composite Materials, **39**:6, 497-524, 2005.

Adjoint Shape Optimization of U-Bend Duct for Pressure Loss Reduction

G. Alessi[1,*], L. Koloszar[1], T. Verstraete[1,2], B. Blocken[3] and J. van Beeck[1]

[1] *von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, B-1640, Belgium*

[2] *Queen Mary University of London, London, E1 4NS, United Kingdom*

[3] *KU Leuven, Leuven, 3000, Belgium*

The pressure loss reduction inside a U-Bends is of crucial importance to increase the performance of cooling systems of gas turbines. The optimization technique proposed is a gradient based shape optimization method and is implemented in an open-source framework. The adjoint method has been used to efficiently compute the sensitivity. An optimum design that assure a minimum pressure loss is obtained via a steepest descent method, satisfying geometrical constraints to account for structural limits.

I. Introduction

Cooling systems of gas turbines are composed by internal channel connected by U-bend passages. Referring to these applications, in most cases the coolant is air bled from the high pressure compressor. A proper design should minimize the coolant mass flow rate, i.e. through a reduction of the pressure loss, maintaining the desired thermal exchange. The U-bends that connect consecutive passages are of paramount importance since they represent regions responsible of high pressure loss. As consequence, design improvements can be obtained through the pressure loss minimization of the U-Bends passage.

The adjoint shape optimization has been chosen in the present work to achieve the described goal. Among the different optimization method existing, the adjoint method is particularly interesting, for its computational cost independent from the number of design variables, opening up a vast design freedom.

II. Adjoint shape optimization in an open-source framework

The optimization will follow the schematic loop shown in Figure 1.

In the continuous adjoint formulation[1] two systems of equations have to be solved: the Navier-Stokes equations and the adjoint equations. The first step of the loop is to solve the Navier-Stokes equations (primal system) to obtain convergent flow variables; successively the convergent assessment is required for the adjoint variables. By solving both systems of equations, a surface sensitivity map can be extrapolated. A surface sensitivity map represents for each and every surface node how the objective function changes with respect to an infinitesimally small normal displacement of this surface node. Based on this information, the body contour has to be moved in the direction normal to the surface itself. In the present work, the geometry movement has been coupled with a mesh morphing solver, allowing automatic successive steps of the optimization process. In order to obtain a smooth surface, the sensitivity has been averaged and normalized. Geometrical constraints have been added to account for structural limits.

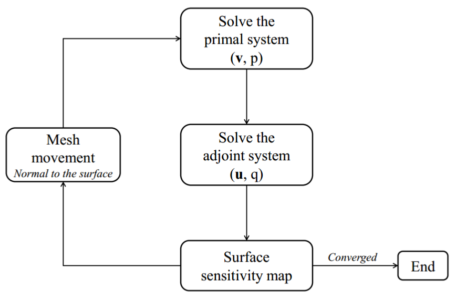


Figure 1. Optimization loop.

The different steps of the optimization loop (solution of the primal and adjoint field, evaluation of the surface sensitivity map, geometry movement and mesh morphing) have been performed by means of the open-source software OpenFOAM.

*PhD student, giacomo.alessi@vki.ac.be, Environmental and Applied Fluid Dynamics Department, Waterloosesteenweg 72 B-1640 Sint-Genesius-Rode Belgium.

III. Results

The first step of the optimization loop regards the evaluation of the flow field inside the U-Bend. The numerical domain is represented by a circular U-Bend of square section (hydraulic diameter D_h=0.075 m), whose geometrical details can be found in Verstraete et al.[2] The experimental tests, Coletti et al.,[3] were performed in atmospheric conditions, considering a bulk velocity of U_0 = 8.4 m/s and a turbulence intensity at the inlet of T.I.=5%.

In order to validate the numerical results, the numerical boundary conditions have to be representative of the experimental campaign. Figure 2 illustrates a comparison of the velocity profile imposed at the inlet of the numerical domain with the experimental one (across the center of the duct). The obtained profile is in good agreement with the experimental one. A zero pressure boundary condition is applied to the outlet.

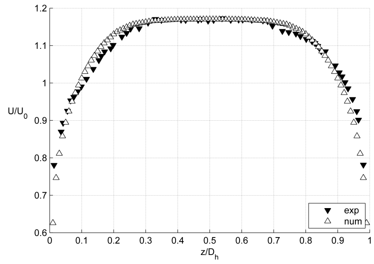


Figure 2. Comparison between the experimental and numerical velocity inlet profiles at the center of the duct.

The numerical simulation has been performed using a structured grid of 342x50x50 elements, assuring a maximum y+ value of 1.63. The Lauder-Sharma low-Reynolds k-ϵ turbulence model has been used. Figure 3 shows the velocity field at the middle plane of the U-Bend. The flow accelerates approaching the bend, it reaches the maximum velocity around the inner wall while it decelerates along the outer wall. The flow starts to separate before the end of the bend and form a separation bubble 1.7D_h long. The magnitude of the reverse flow reaches a maximum value of around 0.45U_0.
The main parameter to compare is represented by a normalized static pressure drop:

$$\Delta P = \frac{P_{s2} - P_{s1}}{\frac{1}{2}\rho U_0^2} \tag{1}$$

The obtained pressure drop is of ΔP=1.04, in good agreement with the experimental value of 1.03$\pm$0.03.

The successive step of the optimization loop regards the evaluation of the adjoint field and the attainment of the surface sensitivity map. The considered cost function is the minimization of the total pressure loss between the outlet and the inlet. Figure 4 illustrates the obtained surface sensitivity map at the middle plane. It indicates the modifications to perform for an improvement of the cost function, in particular: a movement away from the fluid of a positive area and a surface modification towards the fluid of a negative region. Modifications of a zero gradient region have little effect on the cost function.

Finally, the original geometry can be modified accordingly to the surface sensitivity map and the loop is restarted. The optimized shape is attained at the convergence of the loop.

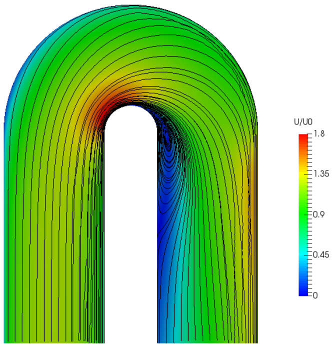


Figure 3. Velocity field at the middle plane.

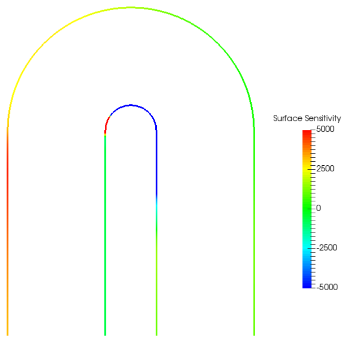


Figure 4. Surface sensitivity at the middle plane.

References

[1]C. Hinterberger and M. Olesen. Industrial Application of Continuous Adjoint Flow Solvers for the Optimization of Automotive Exhaust Systems. *ECCOMAS Thematic Conference - CFD & Optimization*, Antalya, 2011.

[2]T. Verstraete, F. Coletti, J. Bulle, T.V. der Wielen, N. van der Berge and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part I: Numerical Method. *ASME Journal of engineering for Gas Turbines and Power*, 135:pp 051015, 2013.

[3]F. Coletti, T. Verstraete, J. Bulle, T.V. der Wielen, N. van der Berge and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part II: Experimental validation. *ASME Journal of engineering for Gas Turbines and Power*, 135:pp 051016, 2013.

Adjoint based design optimization of a U-bend for minimized pressure losses

Lasse Mueller*
Von Karman Institute for Fluid Dynamics, Turbomachinery Department, Waterloosesteenweg 72, Sint-Genesius-Rode, Belgium

Tom Verstraete† and Jens-Dominick Mueller‡
Queen Mary University of London, School of Engineering and Material Science, Mile end road, E1 4NS, London, UK.

The aim of this paper is to reduce the pressure losses of a u-bend passage of a serpentine cooling channel of a turbine blade. A steady state Reynolds Average density based Navier-Stokes solver is used to predict the pressure losses at a Reynolds number of 40,000. The u-bend shape is parameterized using trivariate BSplines defining directly the volume of the internal passage. The deformations of the shape are controlled by the external control points of the BSpline volume, while the internal control points are repositioned using an elliptic smoothing algorithm to ensure a smooth and regular internal representation of the shape. The sensitivities of the control points with respect to the objective function are computed using a hand-derived adjoint solver and a backward differentiated geometry generation system.

I. Introduction

INTERNAL cooling channels of turbine blades are essential to enable high firing temperatures, and hence high efficiencies. They are characterized by multiple passages of relatively cool air through serpentine ducts. Among the salient features of these cooling passages, the U-bends that connect consecutive passages play a key role by turning the flow 180 degrees while still cooling the outer structure. The aim of this paper is to reduce the pressure loss associated to the flow turning by optimizing the shape of the bend. The U-bend under study is one which has been investigated experimentally at the von Karman Institute for Fluid Dynamics[1,2].

II. Parameterisation

A novel parametrization method has been developed to directly deform the internal volume of the U-Bend. The most classical approach in CAD to define shapes is through a boundary representation method, in which the shape is modeled by its skin using trimmed NURBS surfaces. This method is widely used in different commercial CAD packages and is the standard method adopted in industry to design and optimize shapes. When this representation of the geometry is used to perform further analyses, e.g. CFD computations, a volume grind needs to be generated based on the boundary definition of this CAD model. This process requires to generate internal grid points without a proper description from the CAD model of the volume surrounding the boundary.

In the present work, we use a volume representation of the shape by using trivariate BSplines. This has as advantage that the internal volume points are defined by a unique set *(u,v,w)* of local coordinates, which can be transformed to the Euclidian space. This simplifies significantly the mesh generation of the volume, as now a regular rectangular grid in *(u,v,w)*-space can be transformed to the *(x,y,z)*-space. Figure 1 illustrates the relationship between the *(u,v,w)*-space and the *(x,y,z)*-space, demonstrating that this relationship is controlled by the 3 dimensional net of control points. The position of the control points defining the external skin of the volume are design parameters during the optimization, while the internal control points are repositioned following a design change by an elliptic smoothing algorithm, hence guaranteeing a sufficient level of grid regularity.

Communicating Author: t.verstraete@qmul.ac.uk
*PhD student.
†Visiting Professor.
‡Senior Lecturer.

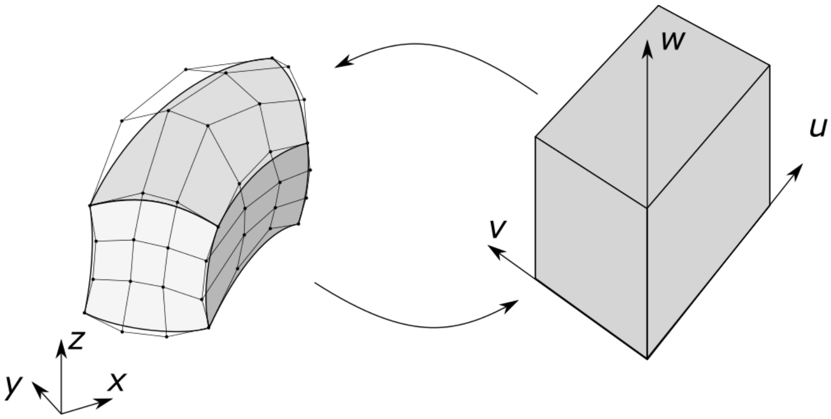


Figure 1. Relationship between the (u,v,w) space and the (x,y,z) space.

III. Results

As a first step a 2D optimization has been performed using bivariate Basis Splines. In Fig. 2 the resulting shape from the optimization is compared with the original shape. A 53% reduction in entropy increase has been achieved.

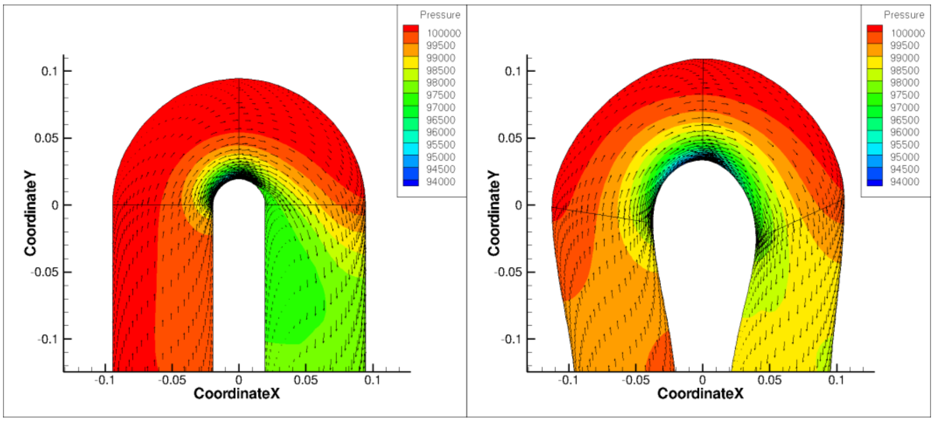


Figure 2. Comparison of original (left) and optimized (right) shape of the u-bend.

Acknowledgements

We gratefully acknowledge the financial support from the European Commission under the project 'MADDOG' (Grant Agreement No. 660759).

References

[1]T. Verstraete, F. Coletti, J. Bulle, T.V. der Wielen, N. van der Berge and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part I: Numerical Method. ASME Journal of engineering for Gas Turbines and Power, 135:pp 051015, 2013.

[2]F. Coletti, T. Verstraete, J. Bulle, T.V. der Wielen, N. van den Berge, and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part II: Experimental validation. ASME Journal of engineering for Gas Turbines and Power, 135:pp 051016, 2013.

U–Bend Optimization on the RBF4AERO Platform

Dimitrios H. Kapsoulis, Evangelos M. Papoutsis-Kiachagias,
Varvara G. Asouti, Kyriakos C. Giannakoglou
National Technical University of Athens, School of Mech. Eng.,
Parallel CFD & Optimization Unit, Athens, Greece

Emiliano Costa
D' Appolonia S.p.A.
Viale Cesare Pavese 305, 00144 Rome, Italy

Marco E. Biancolini
University of Rome Tor Vergata, Dept. of Enterprise Engineering
Via Politecnico 1, 00133, Rome Italy

This work is based on both stochastic and gradient–based optimization methods used to solve industrial optimization problems. The optimization is carried out through the RBF4AERO platform, developed in the framework of the EU–funded project. The stochastic optimization tool of the platform uses Evolutionary Algorithms (EAs) assisted by off–line trained surrogate models, based on the appropriate sampling of the design space. The continuous adjoint developed on the OpenFOAM Toolbox that provides the sensitivity derivatives for gradient–based methods is the second optimization tool available on the same platform. In either method, the design variables stand for the coordinates of points controlling the deformation of the shape to be optimized along with the computational mesh. Shape and mesh morphing is based on Radial Basis Functions (RBFs). Herein, the aforementioned platform is used for the shape optimization of a U–bend for minimum total pressure losses.

I. Introduction

Nowadays, it is common for almost all industries to try to optimize their products, so as to achieve better performance/efficiency. Different optimization methods, stochastic and/or gradient–based ones, have been developed with their own features, advantages and disadvantages.

Evolutionary algorithms (EAs), the most popular stochastic optimization method, can handle any complex/constrained problem using any evaluation tool as a black–box, but at high cost, i.e. a lot of evaluations, for reaching the optimal solution. Among other, surrogate evaluation models are usually coupled with EAs, reducing the optimization turnaround time and making them attractive for industrial applications. On the other side, though computing the gradient using adjoint method requires high effort to develop and maintain the solvers, the optimal solution can be found in a few optimization cycles irrespective of the number of the design variables.

To profit from the advantages of both methods, the RBF4AERO platform[1] implements both EAs and continuous adjoint methods and can be used to cope with industrial scale optimization problems.

II. The optimization tools of RBF4AERO Platform

Apart from the optimization methods, the RBF4AERO platform comes with CFD and FEM solvers and is supported by a GUI which facilitates the set–up of the optimization problem.

In the case of EA–based optimization,[2] a sampling technique (Design of Experiments) is used to select the initial individuals, with which the surrogate model, herein Response Surface Method (RSM), is trained. Then, the EA evolves using the trained RSM as the low–fidelity evaluation tool. The "optimal" solutions found are re–evaluated on the high–fidelity evaluation tool. In the course of the optimization algorithm, the RSM is regularly updated.

On the other hand, the gradient–based method assisted by the continuous adjoint method starts from an initial point on the design space, practically the baseline geometry, and moves towards the "optimal" solutions using the gradient of the objective function. Using adjoint methods,[3] the cost for computing the gradient does not scale with the number of design variables.

In either tool, a morphing tool based on RBFs[4] undertakes the computational domain (including its boundary) deformation, according to the values of design variables. When coupled with the adjoint solver providing the sensitivity maps, the morpher tool additionally computes the grid deformation velocity, i.e. the gradient of mesh coordinates w.r.t. the design variables. The overview of this procedure is shown in figure 1.

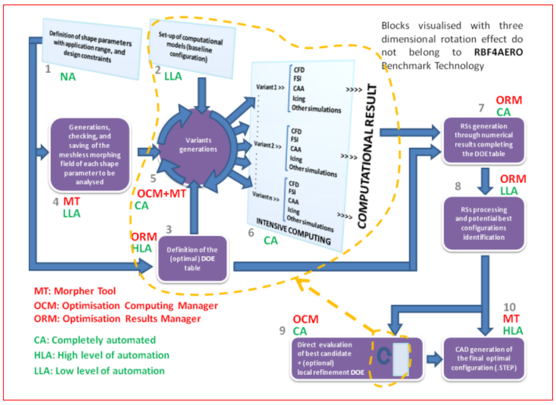


Figure 1. RBF4AERO optimization process

III. Results

The above algorithms included in the RBF4AERO platform are used for the shape optimization of a U–bend, aiming at minimum total pressure losses. The simpleFOAM solver is used along with the Spalart–Allmaras turbulence model. The continuous adjoint method includes differentiation of the turbulence model. Figure 2 presents the flow field at the midspan of the channel for the baseline U–bend geometry along with the control box used for the shape optimization.

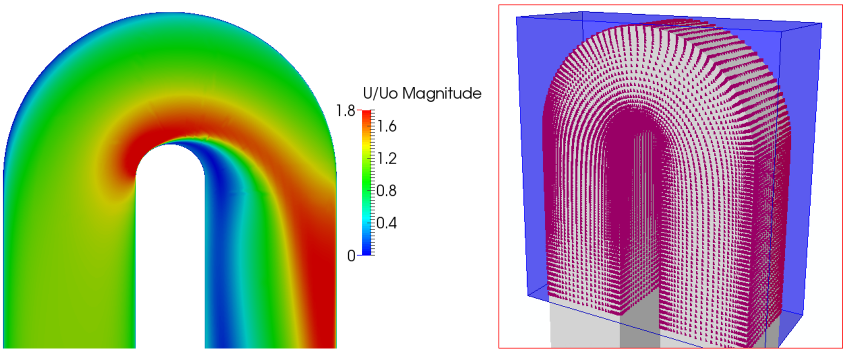


Figure 2. Design of a U–bend. Normalized velocity field in the midspan of the channel for the baseline U–bend geometry (left) and the control box used for the shape optimization (right).

Acknowledgment

This work is financially supported by the RBF4AERO Project, funded by the European Union 7th Framework Programme (FP7-AAT, 2007 - 2013) under Grant Agreement no. 605396.

References

[1]RBF4AERO *http://www.rbf4aero.eu/*,

[2]D.H. Kapsoulis, V.G. Asouti, K.C. Giannakoglou, S. Porziani, Costa, E., Groth, C., Cella, U., Biancolini, M.E. *Evolutionary Aerodynamic Shape Optimization Through the RBF4AERO Platform*, European Congress on Computational Methods in Applied Sciences and Engineering, Crete Island, Greece, 5-10 June 2016.

[3]EM. Papoutsis-Kiachagias and KC. Giannakoglou *Continuous adjoint methods for turbulent flows, applied to shape and topology optimization: Industrial applications*, Archives of Computational Methods in Engineering, DOI 10.1007/s11831-014-9141-9, 2014.

[4]M.E. Biancolini *Mesh morphing and smoothing by means of radial basis functions (RBF): A practical example using fluent and RBF morph*, Handbook of Research on Computational Science and Engineering: Theory and Practice, vol 2:347380.

Shape optimisation with differentiated CAD-kernel for U-bend testcase

Orest Mykhaskiv* and Jens-Dominik Müller
Queen Mary University of London, London, E14NS, UK

Salvatore Auriemma† and Herve Legrand
OpenCASCADE, Guyancourt, 78280, France

Mladen Banovic‡ and Andrea Walther
University of Paderborn, Paderborn, 33098, Germany

A gradient-based optimisation of a parametric CAD model requires the calculation of shape sensitivities, i.e., the derivatives of surface points with respect to the design parameters. Typically, this information is not available within a CAD system and can be obtained by applying Automatic Differentiation (AD) to the CAD sources. This paper demonstrates the differentiated open-source CAD kernel OpenCascade Technology (OCCT) using the AD software tool ADOL-C (Automatic Differentiation by Over-Loading in C++) for the optimisation of pressure loss in a U-bend pipe. The U-bend geometry is parametrised in OCCT and its derivatives are used in CFD optimisation loops.

I. Introduction

To include CAD-models in the shape optimisation workflow one needs to find an effective way to compute CAD sensitivities. One of the strategies in industry is to build perturbed models and calculate shape derivatives with finite differences. This approach results in gradient inaccuracy and possible topology changes due to finite-step displacements.

While hand-differentiation of a complete CAD-system is unlikely to be feasible, the application of automatic differentiation (AD) tools proves to be a way forward. Xu et al.[1] apply AD in the NSPCC approach which uses the NURBS control points in the CAD-native boundary representation (BRep) as degrees of freedom. It works with the CAD-vendor neutral STEP file, although additional constraints need to be imposed to retain the desired continuity between NURBS patches.

Here we demonstrate the application of the AD-tool ADOL-C to the CAD system OCCT in order to enable parametric CAD optimisation. This overcomes the deficiencies of previous methods as there are no topological changes and constraints can be easily embedded directly in the CAD-model.

II. Automatic Differentiation of OpenCascade Technology

ADOL-C[2] is a software tool that uses the operator overloading concept to compute, in forward and reverse mode of automatic differentiation, first and higher derivatives of vector functions that are defined by computer programs written in C/C++. For the purpose of this paper, only one feature of ADOL-C is considered, i.e., the *traceless forward differentiation* - which computes first order derivatives in scalar and in vector mode.

The basis of automatic differentiation by overloading is the concept of `active` variables. All variables that may be considered as differentiable quantities at any part of the program must be of an `active` type - which is called `adouble` in ADOL-C. Therefore, the idea is to replace the declaration types of all relevant `real` variables by `adouble` type. To achieve this, several possible ways of ADOL-C integration into OCCT were considered, but only one was successful so far - the `typedef` approach. This approach is very intrusive because every `double`, even one not needed for differentiation, is replaced by `adouble`, but it is the fastest way of integration.

*Marie-Curie Early Stage Researcher, School of Engineering and Material Science, Mile End Road, London
†Marie-Curie Early Stage Researcher, OpenCASCADE, 1 Place des Frres Montgolfier, Guyancourt
‡Marie-Curie Early Stage Researcher, Institute of Mathematics, Warburger str. 100, Paderborn

III. Parametrisation

The U-bend under investigation, as shown in Figure 1, is a typical internal cooling channel. Its optimisation problem has already been studied[3] where the design parameters are actually the control points of U-bend's B-spline surfaces. In this paper we propose a new parametrisation that is based on a cross-sectional design approach - the lofting. In particular, the part of the U-bend to be optimised is described by N-slices generated along a guiding pathline and approximated with a tool provided by OCCT. Each slice lies on a plane which is orthogonal to the pathline and consists of the 4 Bezier curves forming a closed wire and having in total 12 control points. The position of each control point on the plane is determined by a law of evolution of the control point along the pathline. The laws of evolution are described with B-spline curves. For this reason, the law's control points are the design parameters of the optimisation.

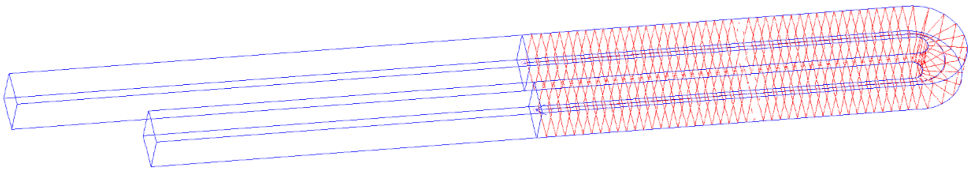
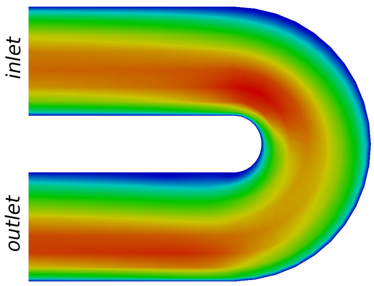


Figure 1: U-bend

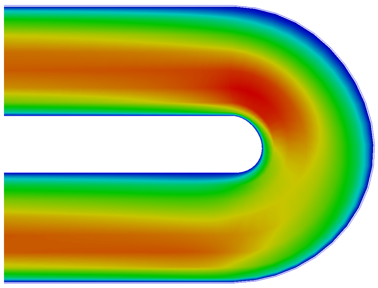
IV. Results

The U-bend described above is subject to aerodynamic shape optimisation to reduce pressure losses between the inlet and outlet legs. For the CFD analysis an in-house solver *mgOpt* was used. It facilitates the geometric multi-grid method and implements the discrete adjoint method using AD. It was coupled with the differentiated OCCT to obtain gradient information driving the optimisation loop.

In Figure 2 the preliminary results of a simpler U-bend geometry optimisation are shown by means of flow velocity. The optimiser managed to suppress the separation bubble formed in the beginning of an outlet leg.



(a) Initial design



(b) Optimised design

Figure 2: Velocity Magnitude

References

[1] S. Xu, J. Wolfram, J.-D. Müller, CAD-based shape optimisation with CFD using a discrete adjoint, *Int. J. Numer. Meth. Fluids*, 74: 153–168, 2013.

[2] A. Griewank and A. Walther, Getting Started with ADOL-C. *Combinatorial scientific computing*, 181 – 202, 2009.

[3] T. Verstraete, F. Coletti, J. Bulle, T. Vanderwielen, and T. Arts, Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - part 1: Numerical Method. *ASME Turbo Expo 2011: Power for Land, Sea and Air*, GT2011-46541.

Geometric Immersed Boundaries (GIB): A New framework for applying boundary conditions in Finite Volume Method

Georgios K. Karpouzas* and Eugene de Villiers
Engys Ltd., London, SW18 3SX, UK

In this paper, a novel method for applying boundary conditions in finite volume method, is presented. In this new framework, standard boundary conditions (Diriclet, Neumann etc.) can be applied on new immersed boundaries which are constructed from existing internal faces and produce identical results to the standard boundaries.

This methodology is powerful in applications with moving parts such as topology optimization, rotating gears, FSI, etc. Currently, in these applications, the mesh motion algorithm moves the boundaries until re-meshing is required. This method, in most industrial applications, is inefficient or unfeasible. Using GIB, the point coordinates of the faces near the interface are snapped on the interface. After the snapping, a group of faces which are located exactly on the interface, is constructed. A new boundary is created based on the interface and boundary conditions are applied. The matrix contributions of each implicit and explicit finite volume operator using GIB and body fitted meshes are the same which guarantees that the results will be identical. The implementation is generic and no additional numerical schemes or executables are required. The method has been developed in the opensource CFD software OpenFOAM®.

The present work has been conducted under the auspices of the ITN Aboutflow FP7 EU project.[1]

I. Results

In this section static and moving immersed boundaries are benchmarked with a body fitted mesh.

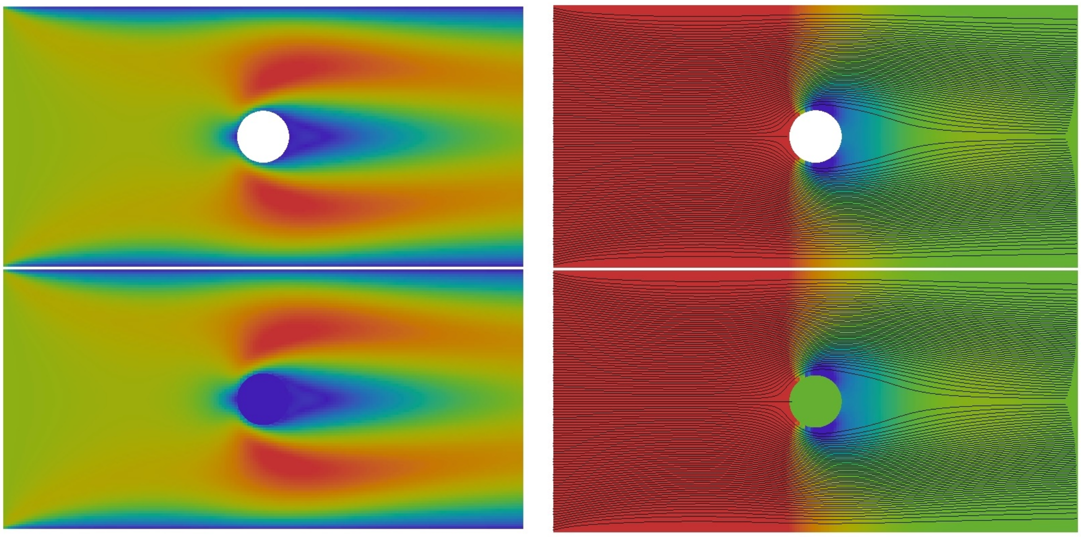


Figure 1: Velocity (left) and pressure (right) fields around a cylinder with classic boundaries (top) and the GIB (bottom). The results in the immersed boundaries case are identical with the bodyfitted approach.

*CFD Software Developer, Engys Ltd.

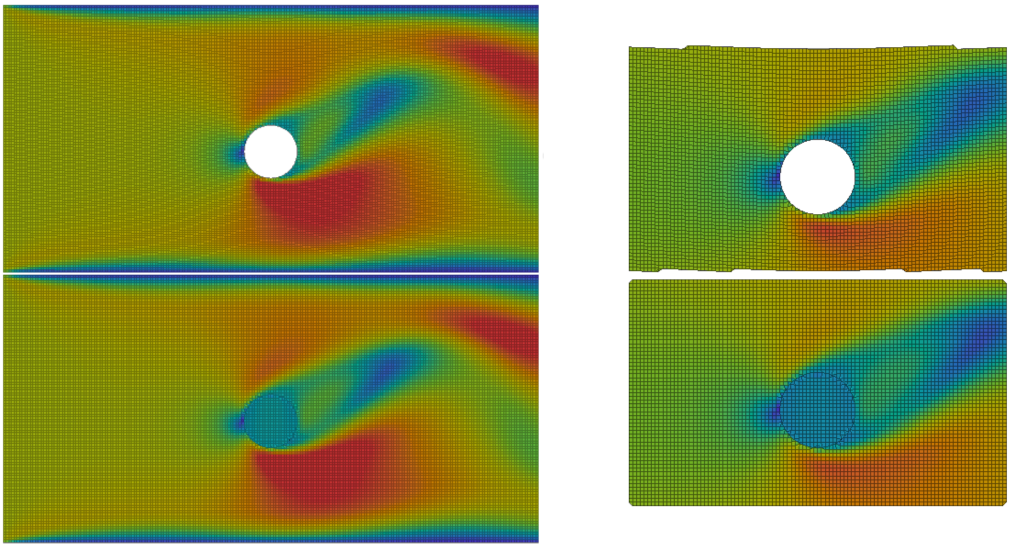


Figure 2: Velocity field around a moving cylinder with classic boundaries (top) and the GIB (bottom).

References

[1]Adjoint based optimization of industrial unsteady flows, *http://aboutflow.sems.qmul.ac.uk*

[2]M. Uhlmann *An immersed boundary method with direct forcing for the simulation of particulate flows,* Journal of Computational Physics, Volume 209, Issue 2, 1 November 2005, Pages 448–476.

[3]Yu-Heng Tseng, Joel H.Ferziger, *A ghost-cell immersed boundary method for flow in complex geometry,* Journal of Computational Physics, Volume 192, Issue 2, 10 December 2003, Pages 593–623.

[4]L. Schneiders, D. Hartmann, M. Meinke, W. Schrder, *An accurate moving boundary formulation in cut-cell methods,* Journal of Computational Physics, 235 (2013) 786–809.

Rigid Motion Mesh Morpher: a robust morphing tool for adjoint-based shape optimization

George Eleftheriou and Guillaume Pierrot

ESI Group, Rungis, 94513 CEDEX, France

A complete discrete adjoint-based shape optimization process necessitates the use of a reliable and robust mesh deformation tool. The Rigid Motion Mesh Morpher is a mesh deformation tool based on the principle of gracefully deforming the mesh in a way that keeps the motion of its parts (groups of nodes) as-rigid-as-possible. This is achieved by minimizing a distortion metric, hence solving the problem of minimization of a mesh deformation energy functional.

I. Introduction

In adjoint based shape optimization problems, there are two ways of dealing with the required changes to the mesh. The first one is regenerating a mesh (re-meshing) based on the new shape, while the second one is adapting the existing mesh (by moving the nodes) to fit the new shape (morphing or mesh deformation). Re-meshing may be time consuming, as it is introduced as a separate step inside the optimization loop, and tedious as it may require user-intervention. It also introduces inconsistencies in the process as the sensitivities are normally supposed to be computed at isoconnectivity. Morphing, on the other hand, is also challenged, mainly by the need to maintain the mesh quality (avoiding distorted and negative cells) while deforming it. Within this context, various mesh morphing techniques have been developed. The Spring Analogy[1] is simple but may suffer robustness issues. Laplacian smoothing[6] is suitable for translation but does not account for rotation. The Linear Elasticity approach[3] does not account for mesh anisotropy and is difficult to implement for general meshes because finite elements are used to solve the equations. Finally, the Radial Basis Functions[5] are promising but computationally heavy as the matrices involved are full, restricting the mesh size and complicating the implementation.

II. Rigid Motion Mesh Morpher

The Rigid Motion Mesh Morpher approach, as is shown in the present study, overcomes the above-mentioned limitations, being more flexible and essentially mesh-less, since it does not require any inertial quantities or cell connectivities related to the mesh. Firstly, the set of surface nodes (or boundary nodes, the ones defining the shape) of the mesh is identified. The prescribed motion of these nodes (their velocities) is known. Then, all nodes are grouped into “stencils” and all the stencils are required to deform in an as-rigid-as-possible way. Thus, for every stencil we define a rotation velocity and a translation velocity. Those, along with the velocities of all volume nodes (or internal nodes), form the set of unknowns. The as-close-to-rigid-as-possible motion is achieved by attempting to minimize a metric representing the difference of the actual deformation from a perfectly rigid motion (simply a translation plus a rotation). The resulting system of equations is solved, using the prescribed motion of the boundary nodes as input to form the boundary conditions.

The aforementioned qualities hold true for as long as the prescribed displacements of the boundary nodes are tiny enough, in order to remain within the linear range of the problem. For larger displacements, subcycling must be used in order to subdivide the prescribed displacements into smaller ones. Hence, a “rigid-motion” history of the stencils is kept at all times during the morphing cycles to compute the extent to which some stencils need to be rigidified.

An improvement upon this concept, is the development of a non-linear variant, namely, the Finite Transformation Rigid Motion Mesh Morpher, that eliminates the need for subcycling and keeping track of the “rigid-motion” history of the stencils for all subcycles. It employs the Polar Decomposition[4] to compute a rotation matrix and bears some similarity to.[2] Finally, there is a significant gain in terms of morphing efficiency and the quality of the resulting mesh.

III. Results

The tool has been tested and benchmarked both as a standalone tool (no optimization case involved) and as a link of an adjoint-based shape optimization toolchain. Mid-size industrial test cases have been run and the results will be presented.

References

[1]Carlo L. Botasso, David Detomi, and Roberto Serra. The Ball-Vertex Method: a New Simple Spring Analogy Method for Unstructured Dynamic Meshes. *Computer Methods in Applied Mechanics and Engineering*, 194:4244–4264, 2005.

[2]Isaac Chao, Ulrich Pinkall, Patrick Sanan, and Peter Schröder. A simple geometric model for elastic deformations. In *ACM Transactions on Graphics (TOG)*, volume 29, page 38. ACM, 2010.

[3]Richard P. Dwight. Robust Mesh Deformation using the Linear Elasticity Equations. In *Computational Fluid Dynamics 2006*, pages 401–406. Springer, 2009.

[4]Nicholas J Higham. Computing the polar decomposition-with applications. *SIAM Journal on Scientific and Statistical Computing*, 7(4):1160–1174, 1986.

[5]Stefan Jakobsson and Olivier Amoignon. Mesh deformation using radial basis functions for gradient-based aerodynamic shape optimization. *Computers & Fluids*, 36(6):1119–1136, 2007.

[6]Z. Su, S. Wang, C. Yu, F. Liu, and X. Shi. A Novel Laplacian Based Method for Mesh Deformation. *Journal of Information and Computational Science*, 7(4):877–883, 2010.

CAD-Free Soft Handle Parameterization for Adjoint-Based Optimization Methods

Athanasios G. Liatsikouras*
ESI Software Germany GmbH, Kruppstr. 90, ETEC H4, 3.OG 45145, Essen, Germany &
National Technical University of Athens (NTUA), P.O.Box 64069, 15710 Athens, Greece

George S. Eleftheriou†
ESI Group, 99 rue des Solets, 94513 Rungis CEDEX, France &
National Technical University of Athens (NTUA), P.O.Box 64069, 15710 Athens, Greece

Guillaume Pierrot‡
ESI Group, 99 rue des Solets, 94513 Rungis CEDEX, France

Mustafa Megahed§
ESI Software Germany GmbH, Kruppstr. 90, ETEC H4, 3.OG 45145, Essen, Germany

The aim of this article is to develop and validate a CAD-Free parameterization tool which is necessary in the Adjoint-based optimization methods in order to perform an optimization cycle. The Rigid Motion Mesh morpher ensures an as-rigid-as-possible motion by minimizing a distortion metric. This morpher lacks of parameterization of the boundaries, so the CAD-Free parameterization tool, which is compatible with this existing in-house mesh morpher, is responsible for the displacements of the mesh in such way that the resulting mesh is a compromise in between the prescribed or target velocities and the smoothness requirements of the mesh motion. In this article, CAD-Free Soft Handle Parameterization for Adjoint-Based Optimization Methods is presented and will be tested in some classical optimization test cases.

I. Introduction

In adjoint based shape optimization problems, morphing (or mesh deformation) is an efficient way (as opposed to re-meshing) to apply the necessary shape changes right after the sensitivities have been computed.

The Rigid Motion Mesh Morpher[1] and its new updated version,[2] is a mesh-less method and tool which gracefully propagates the movement of the boundaries (surface mesh) to the internal nodes of the mesh (volume mesh), ensuring an as-rigid-as-possible motion. This morpher needs to be supplemented with a parameterization of the boundaries in order to conduct the optimization process in a practical way. CAD-free[3] and CAD-based parameterization are the two options. In this paper, CAD-free parameterization will be investigated, because it is versatile as it does not require any connection with a third party platform such as CATIA, Open Cascade etc. Since the output of the discrete adjoint method are the sensitivities computed with respect to the node coordinates and because of the limited resolution in the discretization schemes, it is a common case that the calculated sensitivities include numerical noise. If used directly to morph the mesh, the resulting surfaces might not be acceptable. The CAD-free soft handle parameterization tool, proposed in the present study, aims to keep a rich design space while enforcing smoothness to the shape.

*PhD Student in National Technical University of Athens (NTUA), based in ESI Software Germany GmbH, Athanasios.Liatsikouras@esi-group.com.
†PhD Student in National Technical University of Athens (NTUA), based in ESI Group, George.Eleftheriou@esi-group.com
‡ESI Group. Technology Expert Multiphysics Software Architecture Particle Methods Team Leader Product Development (TPU1), Guillaume.Pierrot@esi-group.com.
§ESI - Manager Center of Excellence - CFD & Multiphysics, Mustafa.Megahed@esi-group.com.

The handles are selected based on an appropriate sampling of the surface nodes, and a target velocity value is defined for each handle. Shape changes are thus driven by minimizing the difference between target and actual handle velocities, whilst ensuring that smoothness requirements are enforced. The efficiency of this approach will be demonstrated on some classical optimization test cases.

II. Results

Preliminary results of the CAD-Free Soft Handle Parameterization tool do show the utility of this tool. It has been implemented in a tipgap test case. In this case there is a gap between a blade and its shell. A displacement is applied to the blade, which includes numerical noise. The Rigid Motion Mesh Morpher and its adaptive parameterization tool are used to deform the blade. Some figures that follow, demonstrate the resulting mesh.

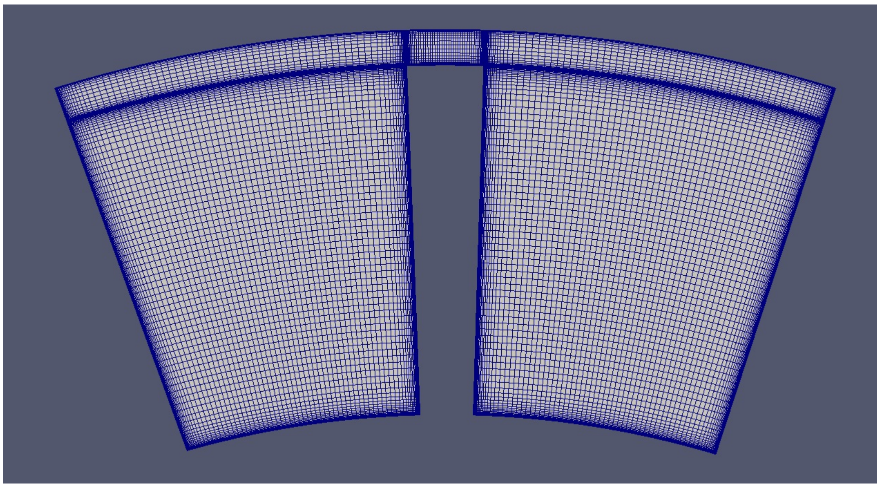


Figure 1. This is the initial mesh of a blade tip test case. In this case, a displacement is applied to the blade, in which numerical noise is included, and in the figures that follow (fig. 2) a comparison between the strict movement of the boundaries and the CAD-Free soft handle parameterization tool is demonstrated.

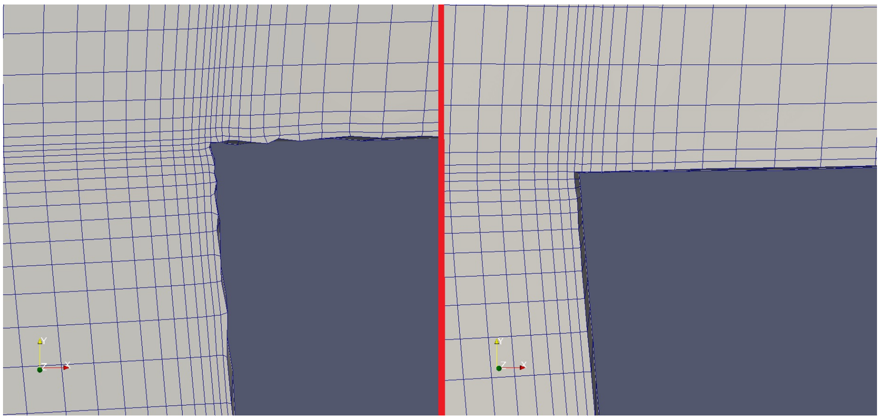


Figure 2. In this figure a comparison is made between the strict movement of the boundaries (left) and the implementation of the CAD-free soft handle parameterization tool (right). The resulting surfaces after the implementation of CAD-free soft handle parameterization do show improvement (A focus is made around the area of interest).

References

[1]George S. Eleftheriou, Guillaume Pierrot, *Rigid Motion Mesh Morpher: A novel Approach for Mesh Deformation*, OPT-i, 2014, Kos Island, Greece 4-6, June 2014

[2]George S. Eleftheriou, Athanasios G. Liatsikouras, Gabriel Fougeron, Guillaume Pierrot, *Finite Transformation Rigid Motion Mesh Morpher*, ECCOMAS Congress 2016, Crete Island, Greece, 5-10 June 2016

[3]Zhixun Su, Shengfa Wang, Chao Yu, Fengshan Liu, Xiquan Shi, *A Novel Laplacian Based Method For Mesh Deformation*, Journal of Information & Computational Science 7: 4 (2010) 877883

A meshless optimised mesh-smoothing framework

Pavanakumar Mohanamuraly* and Jens D. Mueller†

School of Engineering and Material Science,

Queen Mary University of London,

E1 4NS, London, United Kingdom

In this work, we present a least-squares vertex-based mesh quality metric for optimised mesh smoothing, which is inspired from meshless methods. The proposed mesh quality metric requires only point cloud information and does not require mesh topology information. In addition, it is scale, rotational, and translation invariant making it suitable for highly stretched boundary layer type meshes. The proposed quality metric is implemented in the Mesquite mesh optimisation framework.[3] The mesh optimisation is combined with a robust mesh deformation algorithm based on inverse distance weighting[1] for aerodynamic shape optimisation problem. Mesh untangling can be performed using the same mesh optimisation framework by choosing an appropriate quality metric. We show that combining mesh deformation with optimized mesh smoothing can increase the number of design iterations without requiring re-meshing and allows more room for large shape deformation.

I. Introduction

Aerodynamic shape optimisation process typically involves deforming a volume mesh according to a prescribed surface deformation. Robust mesh deformation algorithms based on linear elasticity guarantee deformed meshes with non-negative volumes. But it requires solving a partial differential equation, which is expensive and can have problems with full convergence. The mesh deformation based on inverse distance weighting is a direct interpolation procedure involving simple matrix multiplication. It is found to be quite robust and preserves the quality of the mesh. Mesh deformation methods in general yield meshes of lower quality than the original undeformed mesh. This limits (i) the number of optimisation steps one can run without re-meshing and (ii) the quality of the numerical results. Optimised mesh-smoothing is typically done after the volume mesh deformation to alleviate these problems. Mesh deformation methods like spring analogy or inverse distance weighting can lead to tangled meshes. Mesh smoothing is typically employed to untangle the mesh. An inexpensive mesh smoothing method is the Laplace smoothing[2], which moves the free vertex to the geometric centre of its incident vertices. But the Laplace smoothing does not guarantee improvement in element quality. In reference [2], the authors alleviate this problem by selectively applying the Laplace smoothing to mesh nodes giving rise to the smart Laplacian method.

Guaranteed improvement in mesh quality is obtained using optimisation-based[2-3] approach to mesh smoothing. In this approach, a quality metric such as element angle, skewness, aspect-ratio, etc is chosen as an objective function. The mesh nodes are perturbed to achieve optimal values of the chosen objective function. In practice, the link between an element based objective function and mesh node is not immediate. One has to resort to non-differentiable functions like *min/max* to translate the element measure to the node, which can cause stalling of convergence. We seek a mesh quality metric directly based on nodes and independent of element information, which we present in the next section.

II. Meshless node-based mesh quality metric

Let $\mathbf{x}_i \in \mathcal{T}(\mathbb{R}^N)$ is a vertex in a triangulation $\mathcal{T}(\mathbb{R}^N)$ and $\mathbf{x}_j \in \mathcal{C}(\mathbf{x}_i)$ be the edge neighbours of vertex $\mathbf{x}_i$ connected by the edge vector $\mathbf{r}_{ij} = \mathbf{x}_j - \mathbf{x}_i$. Note that the indices run as $j = 1, ..., m$ and $i = 1, ..., n$. Where, m is the degree of the incident edges to vertex i and n is the total number of vertices in the

*Marie Curie Fellow, School of Engineering and Material Science, Queen Mary University of London

†Senior Lecturer, School of Engineering and Material Science, Queen Mary University of London

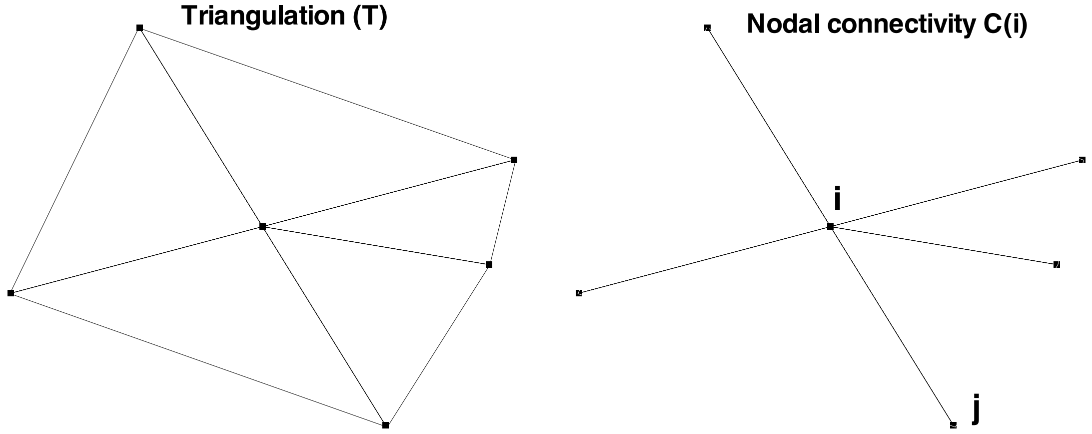


Figure 1. Mesh triangulation and nodal edge neighbour connectivity (solid line)

triangulation $\mathcal{T}(\mathbb{R}^N)$. Let us try to fit a first order polynomial over the neighbourhood of point i, which has the following form,

$$f - f_i = a(x - x_i) + b(y - y_i) + c(z - z_i) = \mathbf{X}\boldsymbol{a} \tag{1}$$

Only three unknowns are required to exactly determine the coefficients $\boldsymbol{\xi}$. But the number of neighbours of the point i are chosen such that, they are more than three, leading to an over-determined system.

$$\mathbf{X}\boldsymbol{a} = \boldsymbol{\Delta}\mathbf{f} \tag{2}$$

Note that we use the notation $\mathcal{C}(i) : \mathbf{x}_j \in \{\mathbf{x}_{j_1}, \mathbf{x}_{j_2}, ..., \mathbf{x}_{j_m}\}$ and $\mathbf{x} = \{x, y, z\}$ in $\mathbb{R}^3$. If we introduce a weighting function $\mathbf{W} = diag\,[w_1 \quad w_2 \quad ... \quad w_m]$ into the equation and convert to the normal form,

$$\boldsymbol{a} = \left(\mathbf{X}^T\mathbf{W}\mathbf{X}\right)^{-1}\mathbf{X}^T\mathbf{W}\boldsymbol{\Delta}\mathbf{f} = \mathbf{M}\boldsymbol{\Delta}\mathbf{f} \tag{3}$$

$$\mathbf{M} = \begin{bmatrix} a_1 & a_2 & ... & a_M \\ b_1 & b_2 & ... & b_M \\ c_1 & c_2 & ... & c_M \end{bmatrix} \tag{4}$$

The matrix $\mathbf{M}$ has a special geometric interpretation. It's column vectors point along the direction of the edges forming the node stencil. For a bad-connectivity they start deviating from the edge vector. Praveen[4] observed that the column vectors of $\mathbf{M}$ deviate more from the actual edge vector, when the edges subtended are highly skewed or have small angles. Ideally a good stencil should have both the column vectors of $\mathbf{M}$ and edge vectors parallel to each other. Any deviation from this parallelism signifies a degradation of mesh quality. We define a mesh metric $\delta_p = \sum_j \delta_{p_j}$ to measure the mesh quality at a node, where δ_{p_j} is given in equation 5. For an ideal mesh distribution around a node $\delta_p = 0$ or a positive quantity ($\delta_p > 0$) for any deviation from the ideal.

$$\delta_{p_j} = |\mathbf{a_j}| - \mathbf{a_j} \cdot \frac{(\mathbf{x_j} - \mathbf{x_i})}{|\mathbf{x_j} - \mathbf{x_i}|} \tag{5}$$

The proposed metric is implemented in Mesquite framework, which comes packaged with a variety of optimisation algorithms. In this work we present results from the steepest descent algorithm.

References

[1] L. Uyttersprot, *Inverse Distance Weighting Mesh Deformation*, Masters thesis, TU Delft, 2014.

[2] Freitag, L., A., and Knupp, P., M., *Tetrahedral mesh improvement via optimization of the element condition number*, Int. J. Numer. Meth. Engng, vol. 53, pp:1377-1391, 2002.

[3] M. Brewer, L. Diachin, P. Knupp, and D. Melander, The Mesquite Mesh Quality Improvement Toolkit, p239-250, proc. 12th International Meshing Roundtable, 2003.

[4] Praveen, C., *Development and application of kinetic meshless method for Euler equations*, PhD Thesis, Indian Institute of Science, Bangalore, 2004.

Applications of the continuous adjoint method to car aeroacoustics

Nikolaos Magoulas * and Michael Hartmann †

Volkswagen A.G., Wolfsburg, 38436, Germany

Applications of the adjoint method in the automotive industry have experienced a rapid growth over the last years for the field of Computational Fluid Dynamics (CFD). The majority of these applications involve either the topology optimization of internal flows (e.g. cabin ventilation ducts, engine intake ports) or the shape optimization of external flows (e.g. drag and lift coefficients). However, CFD has been recently considered for a new field of applications, that of aeroacoustics problems. The wind-generated noise perceived by the passengers of a vehicle impacts negatively the comfort levels inside the cabin. Thus, aeroacoustics methods and procedures have been developed in order to make it possible to calculate the noise levels experienced inside the vehicle's cabin and then improve the aeroacoustics performance of the vehicles. The great potential of the adjoint method in gradient-based optimization and the experience gained over the previous years in the automotive sector, has led the efforts to couple the developed aeroacoustics processes with the adjoint method. Such an example, of coupling an aeroacoustics process with the adjoint method is presented here.

I. Introduction

The advancements in the computational power of computers have led in an ever increasing application of Computational Fluid Dynamics (CFD) in the automotive sector for improving the aerodynamic performance of a vehicle and reducing its development cycle. The negative impact of aerodynamic drag in vehicles fuel consumption has been the main driving factor for improved designs. However the next frontier for CFD is focused on passengers comfort; this time with the development of methods, which can predict the flow-induced noise levels. The aeroacoustic noise generated from the external shape of a vehicle has a great effect on the noise levels experienced inside the passengers cabin, thus requiring the development of tools to improve the acoustic performance of a vehicle.

The adjoint method has played an essential role in the shape optimization for external and internal flows and has been established as the method of choice in the automotive industry. However aeroacoustics problems cannot be tackled with a steady state adjoint method since they are inherently unsteady, based on time fluctuations of the flow fields and more importantly expressed in frequency domain. With the recent development and availability of the unsteady adjoint method at Volkswagen's Group Research[1] , aeroacoustics problems can now be addressed.

In this paper, acoustic analogy methods are considered for the prediction of aeroacoustic noise, with an integral method used to form the basis of an optimization loop,[2] . The method is applied on a test case of a simplified geometry with a sharp edge in order to introduce a separation point and broadband noise effects. The flow simulation is based on an incompressible DES approach and an objective function is formed, which quantifies the high frequency noise levels emmitted from a surface and perceived by a microphone. Thereafter, the unsteady adjoint method is applied in order to compute the gradient of the objective with respect to the design variables, which then drives the optimization loop.

All of the developed methods and simulations were implemented and conducted within the OpenFOAM® software package.

II. Aeroacoustic processes and the Kirchhoff Integral Method

There has been a long history of acoustic analogy methods used in conjunction with CFD in order to predict the flow-induced noise generated in different conditions. Amongst them, the Kirchhoff Integral

*PhD Candidate, Group Research, Letter Box 1777, K-GERFG/V, - National Technical University of Athens, Iroon Polytechniou 9, Polytechnioupolis Zografou, 157 80 Athens, Greece

†Aeroacoustics Senior Scientist, Group Research, Letter Box 1777, K-GERFG/V

(KI), which has been most notably used for the prediction of helicopter noise. Recently, the method has been used to predict the noise generated from the mirror of a moving vehicle, which is then propagated inside the vehicle cabin and perceived by the driver[2] .

On the other hand, the adjoint method has proved to be a robust optimization tool and has been used extensivelly to conduct gradient-based optimization in applications of CFD in the automotive industry[3] . It is therefore promising to combine the experience and knowledge acquired from the optimization of aerodynamics with the adjoint method together with the afformentioned aeroacoustics process chain.

This aeroacoustics process chain consists of 4 major steps:

1. An incompressible DES calculation of the external flow of the vehicle.
2. The acoustic pressure radiated from the mirror on the driver's window is calculated using the KI method.
3. The hydrodynamic and acoustic pressure on the driver's window computed from steps 1 and 2 are combined and provided as an input to compute the structural vibrations of the window.
4. The radiation of noise from the vibrations of the window inside the passanger cabin is calculated for some predefined locations and the noise levels are evaluated at these positions.

Since there had been no prior experience with the adjoint method for steps 3 and 4 (even though the issue has now been addressed, see[4]), a simplified process for the noise generated from the mirror and radiated towards the driver is derived from steps 1 and 2. The unsteady adjoint method for incompressible fluids is then used to tackle the optimization problem defined by this process, as it is presented in this work.

III. Results

A test case is defined and an incompressible DES simulation is used to compute the flow field. Then the KI method is used to compute the acoustic pressure emmited from the geometry towards an array of microphones. The objective function is defined as the sound pressure levels received by the microphones over a range of frequencies. Following that, the unsteady adjoint method is used to compute the flow shape sensitivities, which are then combined with the geometric sensitivities of the KI and are used to perform a morphing step. The objective function is reevaluated for the morphed geometry and a discussion follows comparing the results of the different geometries.

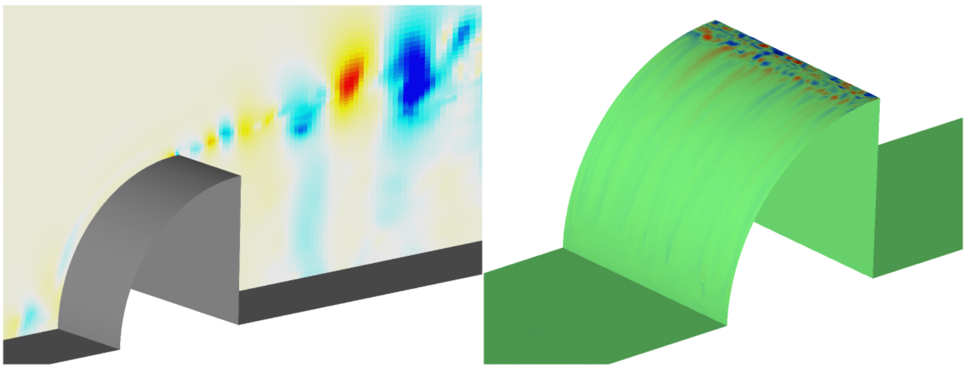


Figure 1. Left: Pressure fluctuations on cutting plane of test case. Right: Shape sensitivities of test geometry.

Acknowledgement: This project has received funding from the European Unions Seventh Framework Programme for research, technological development and demonstration under grant agreement no [317006].

References

[1]N. Magoulas, C. Othmer, E.M. Papoutsis-Kiachagias, K. Giannakoglou: Flow control sensitivities for vehicle aerodynamics. *11th. World Congress on Computational Mechanics (WCCM XI)*, Barcelona, Spain, July 20-25, 2014.

[2]A. Kabat vel Job: Prediction of vehicle cabin noise based on time domain methods. *SAE NVH*, Grand Rapids, June 6, 2015.

[3]C. Othmer: Adjoint Methods for car aerodynamics. *Journal of Mathematics in Industry*, Vol. **4**, Issue 6, 2014.

[4]C. Kapellos, M. Hartmann: A continuous adjoint approach for vehicle interior noise reduction. *ECCOMAS Congress 2016*, Crete Island, Greece, June 5-10, 2016

Unsteady/Steady Continuous Adjoint Method Using a Block Coupled Solver in OpenFOAM: Application on the Drivaer Vehicle

Christos K. Vezyris
Evangelos M. Papoutsis-Kiachagias
Kyriakos C. Giannakoglou
National Technical University of Athens, School of Mech. Eng.,
Parallel CFD & Optimization Unit, Athens, Greece

In this work, shape optimization is performed using the unsteady and steady continuous adjoint method. An implicit numerical solver, programmed in OpenFOAM, for both the Navier-Stokes and the corresponding continuous adjoint equations, is used.

The implicit, block-coupled solver is based on the pseudo-compressibility approach and foam-extend-3.1 is used as the programming environment. The block-coupled solver computes the solution for the flow variables simultaneously, leading to faster convergence and an implicit treatment of the numerically stiff Adjoint Transpose Convection (ATC) term.

A drag minimization problem is considered. The Drivaer vehicle is studied, which is an AboutFlow Adjoint Optimisation Benchmark test case. The objective function is the drag minimization by optimizing the side view mirror.

I. Introduction

The lack of a pressure term in the continuity equation makes the numerical solution of the incompressible Navier–Stokes flow equations challenging. Commonly, using appropriate transformations a Poisson like equation is derived to allow the pressure calculation. OpenFOAM©, among other CFD packages, uses the SIMPLE algorithm to numerically solve the momentum and continuity equations (cast in the form of a Poisson equation) in a segregated manner. An advantage of this approach is the overall low memory requirement[2].

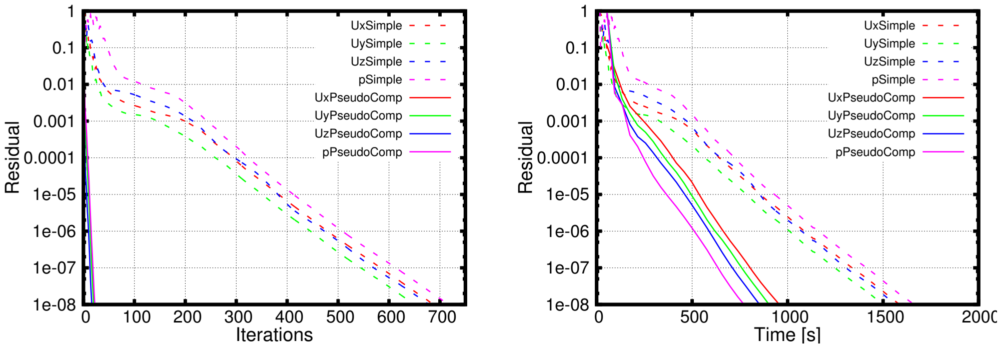
The simultaneous numerical solution that implicit block-coupled solvers undertake, promise faster convergence and lower total computation time. Moreover, potentially greater stability could arise by block-coupled solvers, though in the expense of increased memory requirements.

II. Development of an Implicit Block Coupled solver in OpenFOAM

The physical decoupling between the continuity and momentum equations is overcome by implementing the pseudo-compressibility approach[1]. Both the Navier–Stokes and the corresponding adjoint equations are discretized using the Roe flux scheme. The implicit, block-coupled solver is programmed in foam-ext-3.1 environment. The existing block matrix infrastructure is used, though altered when needed and in conjunction with developed code which allows the optimization process to be conducted.

III. Results

The developed solver is first tested in steady state problems. It is compared with the standard, Navier–Stokes solver for incompressible flows which exists in foam-extend-3.1. A 3D S-bend duct, being among the EU-funded AboutFlow project benchmark test cases, is chosen for demonstration purposes. The computational mesh consists of 446×10^3 cells and the Reynolds number is $\sim$400. In figure 1a and 1b the comparison is illustrated. The number of iterations needed by the block-coupled solver to converge is approximately 25, as seen in figure 1a, while the segregated solver requires approximately 680 iterations. In figure 1b the total computational time for each solver is shown. It is observed that the speed-up resulted by the block-coupled solver is $\sim 2.2\times$.



(a) Number of iteration.

(b) Computational time needed.

Figure 1: Developed block-coupled solver compared with the segregated SIMPLE algorithm based solver. The test case is a 3D, S-bend duct.

References

[1] A. J. Chorin. A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics*, 135(2):118 – 125, 1997.

[2] M. Darwish, I. Sraj, and F. Moukalled. A coupled finite volume solver for the solution of incompressible flows on unstructured grids. *Journal of Computational Physics*, 228(1):180 – 201, 2009.

Aerodynamic Optimization of the TurboLab Stator: A Comparative Study between Conventional and Adjoint-based Approaches

Ilias Vasilopoulos*, Peter Flassig† and Marcus Meyer‡
Rolls-Royce Deutschland Ltd & Co KG, Blankenfelde-Mahlow, D-15827, Germany

Kyra Reiche§
Brandenburg University of Technology, Cottbus, D-03048, Germany

The goal of this work is to develop a CAD-based aerodynamic optimization workflow for the application on the TurboLab Stator benchmark. Two different approaches are examined and compared: (i) A conventional gradient-free optimization procedure which makes use of 2D and 3D CFD results and constructs a response surface to obtain the optimum solution and (ii) an adjoint gradient-based optimization process, where the adjoint derivatives w.r.t. the design parameters are calculated by linking the adjoint surface sensitivity map with the model's design parameter sensitivities. The latter are computed using finite-differences between discrete representations of the CAD geometries. Both methods are applied to the turbomachinery test case, where two objective functions and three operating points are considered. Moreover, manufacturing constraints have to be incorporated in the process chain, which makes the case even more challenging. Since the optimization output is already in CAD format, there is no requirement for a back-to-CAD step and the optimum blade can be straightforwardly manufactured.

I. Introduction

For turbomachinery design, adjoint-driven CFD design offers an efficient approach to improve the aerodynamic performance of the engine gas path and its containing components, like rotor and stator blades [1]. Especially for high ratios between number of design parameters such as blade chord length or blade's thicknesses and number of aerodynamic design criteria such as efficiency, surge margin or capacity, adjoint-based numerical optimization shows clear benefits against classical search strategies in terms of computational effort. This has been demonstrated by a number of publications, such as Ref. 2-4. However, large challenges related to the industrial applicability of adjoint-based methods are not sufficiently resolved yet.

Especially for a CAD-centric approach it is of great importance to have universal and robust methods in place to link the adjoint-based CFD results directly to existing and approved design parameterizations [5,6]. This enables both error-free mapping of the adjoint gradient results to CAD and fast and explicit assessment of design parameter sensitivities by the design engineers. Furthermore, the adjoint method has to be robust and stable for realistic flow characteristics, such as flow separation or instabilities in wakes and vortices [7].

For these reasons, this paper summarizes activities which have been conducted in an industrial environment for the third test case from the "About Flow Adjoint Optimization Benchmark". Aim of the test case is to optimize the 3D shape of the TurboLab Stator from the Technical University of Berlin with respect to aerodynamic criteria. Specifically, the TurboLab Stator is a compressor stator in a one row rig with subsonic flow but high degree of turning [8].

II. TurboLab Stator Optimization

To perform optimization in an automated manner, a proper design parameterization is required. In general, different design parameterizations exist for a compressor blade. However, for the present investigation the approved and powerful classical aerofoil build-up in conjunction with a 3D profile stacking approach is used [9]. More specifically, this parameterization utilizes a non-dimensional camber-line angle distribution, from which

* PhD Student, Engineering Technology – Design Systems Engineering, ilias.vasilopoulos@rolls-royce.com.
† Development Engineer, ET-DSE, peter.flassig@rolls-royce.com.
‡ Development Engineer, ET-DSE, marcus.meyer@rolls-royce.com.
§ Student, Chair of Engineering Mechanics and Vehicle Dynamics, kyra.reiche@tu-cottbus.de.

the camber line is derived, and a non-dimensional thickness distribution. The 2D profile contour for different radial blade heights is obtained by adding thickness values perpendicular to the camber line. A 3D blade shape is generated by a proper stacking definition using axial and circumferential shifts along a defined line in the radial direction [10].

An automated process has been developed and validated to (i) compute the adjoint derivatives w.r.t. each design parameter which is part of the classical aerofoil build-up and (ii) perform gradient-based optimization according to the test case description. In parallel to this purely adjoint-based shape optimization, a conventional direct optimization has been performed, followed by another adjoint-based optimization which uses as a starting point the optimized blade from the conventional search. The results of the three different strategies are compared regarding the aerodynamic performance and required computational cost.

The conventional direct optimization procedure consists of two main steps. Firstly, extensive purely aerodynamic 2D section optimizations are performed for different radial blade heights, where Mises [11] is used to predict the blade-to-blade flow characteristics. Subsequent to this, an optimal stacking line is determined using 3D CFD and emulation-based optimization, where the Rolls-Royce in-house solver HYDRA [12] is used to predict the 3D flow.

III. Results

Preliminary results of the classical 2D-based optimization clearly show performance improvements compared to the reference design, Fig. 1. Especially the turning requirement has been improved by a huge amount. For the adjoint-based optimization, all required derivatives w.r.t. the 176 design parameters have been computed and validated against finite differences for the baseline design. Fig. 2 compares the adjoint sensitivity maps for the baseline, a preliminary unstacked 2D-optimized blade and a preliminary adjoint-based blade after the first optimization step.

From the stacking point of view, the manufacturing constraints from the test case need to be considered carefully. During preliminary studies, it turned out that the blade fixture holes break the blade surface even for small sweep and lean stacking definitions in the vicinity of the hub and tip regions. Therefore, an automated process has been implemented to maximize the stacking parameter bounds before the actual optimization. Using this approach, the aforementioned manufacturing constraint can be guaranteed a priori.

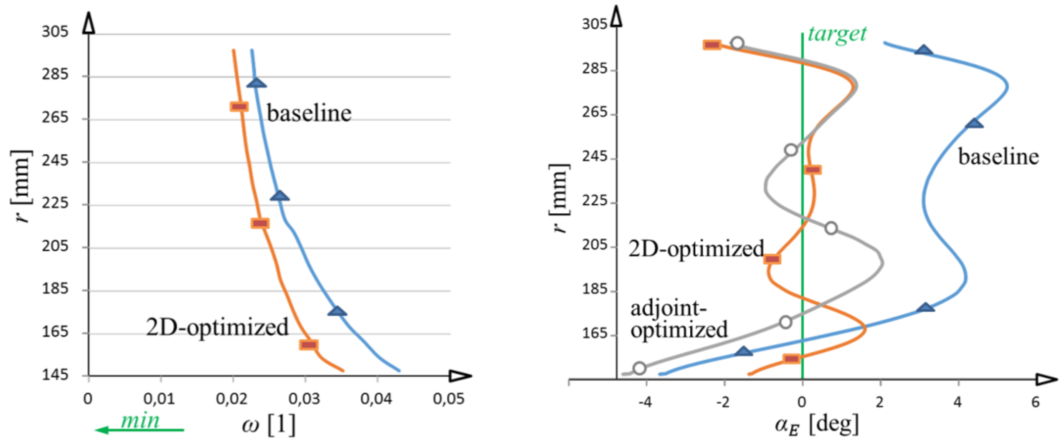


Figure 1: Mises-based 2D CFD losses for baseline and 2D-optimized blade (left) and circumferentially averaged Hydra-based 3D CFD exit whirl angles for baseline, unstacked 2D-optimized blade and preliminary adjoint-optimized blade (right)

Constrained Multi–Objective Optimization of the TU Berlin TurboLab Stator using Continuous Adjoint

F. Gagliardi[*], K. T. Tsiakas[†], X. S. Trompoukis[‡] and K. C. Giannakoglou[§]

National Technical University of Athens

School of Mech. Eng., Lab. of Thermal Turbomachines, Parallel CFD & Optimization Unit

Athens, Greece

An adjoint–based optimization study carried out on the benchmark test–case TU Berlin TurboLab Stator of the 11th international conference on Numerical Optimisation methods for Engineering Design (ASMO UK/ISSMO/NOED2016) is presented. Gradient–based non–linear programming by the Method of Moving Asymptotes (MMA), is used to produce a new stator shape with improved aerodynamic performance. The computational cost is made independent of the number of design variables using the continuous adjoint to compute flow sensitivities. A differentiated in–house turbomachinery row parameterization software is included in the loop. The design variables are NURBS curve coefficients, used as input of the parameterization, which ensure smooth shape changes during the optimization, despite the great number of degrees of freedom. In order to reduce the optimization turnaround time the in-house primal and adjoint solvers are implemented to run on a cluster of Graphics Processing Units (GPUs). For the same reason, costly re–meshing steps are avoided by using Radial Basis Functions (RBFs) to deform the grid accordingly to the updated shape of the flow domain in each cycle of the optimization loop.

I. Introduction

The primary objective of this abstract is to present a workflow for the parameterization and constrained optimization of turbomachinery rows. The test case is a stator in a measurement rig at the TU Berlin in the TurboLab at the Chair for Aero Engines.

II. Optimization Strategy

The stator has been re–parameterized through an in-house turbomachinery row parameterization software, which has been differentiated to support the gradient–based optimization. This strategy allows to maintain a CAD representation of the row during the whole process.

Constraints are specified on the geometry and they are treated as either bounds to the design variable values or constraint functions of the optimization problem. Since two objective functions are considered, a set of non–dominated solutions is sought.

The flow model is based on the Navier–Stokes equations for incompressible flows, using the pseudo–compressibility approach introduced by Chorin.[1] The continuous adjoint method for incompressible flows, with a fully differentiated turbulence model, is used.[2] Both primal and adjoint solvers are implemented on a cluster of NVIDIA Graphics Processing Units (GPUs).[3]

Geometric sensitivities $\delta\mathbf{x}/\delta\mathbf{b}$, which stand for the ratio of boundary displacements $\delta\mathbf{x}$ over the corresponding variation in any of the CAD parameters $\delta\mathbf{b}$, are computed by differentiating the parameterization software. Based on the chain rule of eq. 1, these are combined with the gradients of the objective functions $\mathcal{F}_i$ with respect to (w.r.t.) the displacements of the blade or casing nodes $\delta\mathbf{x}$, as computed by the adjoint method. The outcome is the gradient of the objective functions w.r.t. the variation of any CAD parameters

[*]PhD student, fl.gagliardi@gmail.com.
[†]PhD student, tsiakost@gmail.com.
[‡]Postdoctoral researcher, xeftro@gmail.com.
[§]Professor, kgianna@central.ntua.gr.

$$\frac{\delta \mathcal{F}_i}{\delta \mathbf{b}} = \frac{\delta \mathcal{F}_i}{\delta \mathbf{x}} \frac{\delta \mathbf{x}}{\delta \mathbf{b}} \tag{1}$$

which can be used to change the shape of the stator and casing during the optimization. In fig.1 an example of adjoint and geometric sensitivities is shown.

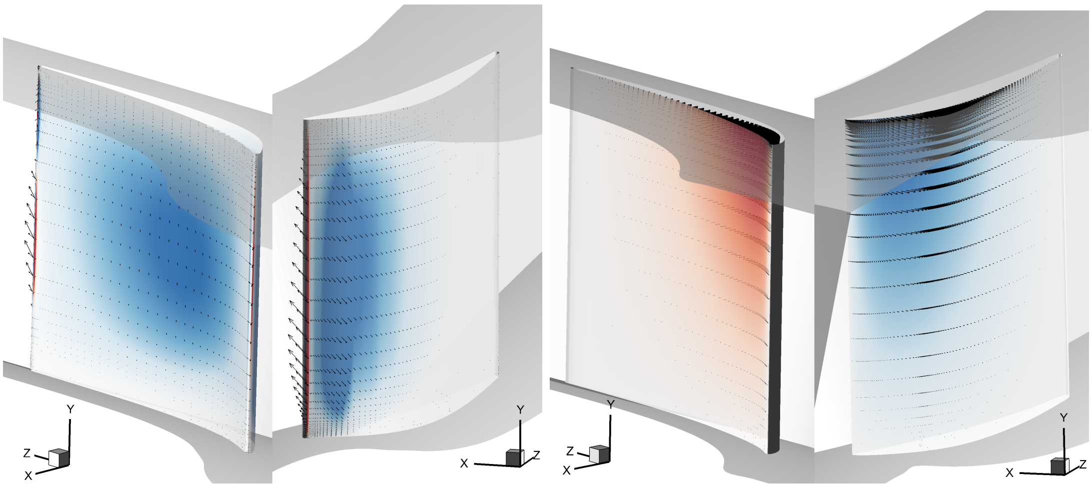


Figure 1. Adjoint sensitivities $\delta\mathcal{F}_i/\delta\mathrm{x}$ of the blade w.r.t. one of the objective functions. The blade is coloured based on the adjoint sensitivities in the normal direction $\delta\mathcal{F}_i/\delta\mathrm{x}$ n, whereas arrows represent the actual adjoint sensitivities $\delta\mathcal{F}_i/\delta\mathrm{x}$ (left). Geometric sensitivities $\delta\mathbf{x}/\delta b$ of the blade w.r.t. one of the design variables. The stator blade is coloured according to the geometric sensitivities in the normal direction $\delta\mathbf{x}/\delta b_i\mathbf{n}$, whereas arrows represent the actual geometric sensitivities $\delta\mathbf{x}/\delta b_i$ (right). For both figures, red colour denotes inwards (towards the solid) displacement, whereas blue outwards.

In order to adapt the mesh to the new shape at each optimization cycle RBFs are employed. With the RBF model the deformation is treated as a scattered data interpolation where surface node displacements are smoothly interpolated at the internal nodes.[4] The new surface grid, which corresponds to the new geometry, is obtained by inverting and displacing nodes in the NURBS parametric space (u, v). Special care is taken in case of trimmed surfaces.

The optimization is driven by the MMA: in this method, in each cycle of the optimization, a convex subproblem approximating the objective function is generated. The "moving asymptotes" are upper and lower bounds defined on each design variable, used to regulate the convergence of the method.[5]

III. Acknowledgement

This research was funded from the People Programme (ITN Marie Curie Actions) of the European Union's H2020 Framework Programme (MSCA-ITN-2014-ETN) under REA Grant Agreement no. 642959 (IODA project). The first author is an IODA Early Stage Researcher.

References

[1] Chorin, A., *A numerical method for solving incompressible viscous flow problems*, Journal of Computational Physics, 2–1, 12–26, New York, 1967.

[2] Papoutsis-Kiachagias, E. M. and Giannakoglou, K. C. *Continuous adjoint methods for turbulent flows, applied to shape and topology optimization: Industrial applications*, Archives of Computational Methods in Engineering, Springer, Athens, 2016.

[3] Asouti,V.G., Trompoukis, X. S., Kampolis and Giannakoglou, K. C. *Unsteady CFD computations using vertex–centered finite volumes for unstructured grids on Graphics Processing Units*, International Journal for Numerical Methods in Fluids, 67(2), 232-246, 2011.

[4] Buhmann, M., *Radial basis functions: theory and implementations*, Cambridge University Press, Cambridge, 2003.

[5] Svanberg, K., *The method of moving asymptotes – A new method for structural optimization*, International Journal for Numerical Methods in Engineering, 24, 359-373, Stockholm, 1987.

Multiobjective Optimisation of a Compressor Stator using a 3D B-Spline Parameterisation

Sam Duckitt and Chiara Bisagni

Delft University of Technology, Delft, 2629 HS Netherlands

Shahrokh Shahpar

Rolls-Royce plc., Derby, DE24 8BJ, United Kingdom

With ever stricter emissions targets and a relentless demand for efficiency improvements, optimisation has become an indispensable tool in the design of gas turbine engines. Traditionally most studies focus on increasing the performance of fan, compressor and turbine blades but it is now common to find optimisation applied to secondary air systems in an effort to further improve overall engine efficiency. Wherever this optimisation is performed in the engine, it inevitably becomes a multidisciplinary endeavor. Compressor blades for example are required to have high aerodynamic efficiency but must also withstand considerable static and dynamic structural loads. Arriving at an optimal design is therefore very challenging since satisfying these objectives results in conflicting requirements.

In this study we focus on a multiobjective optimisation of the TU Berlin TurboLab stator blade. The first objective is to minimise total pressure loss between CFD inlet and outlet whilst keeping the mass flow at $9.0 \pm 0.1kg/s$. The second is to minimise the flow angle deviation at the CFD outlet from the axial direction. Three operating points are accounted for, where the inlet whirl angle is allowed to vary $\pm 5°$ from its nominal value of $42°$. This leads to a multipoint optimisation problem where a weighting of 50% is assigned to the nominal operating condition and 25% to the two off design points. There are also a number of mechanical constraints which will be described during the presentation.

A novel approach has been developed to parameterise the blade with B-spline control points. The idea is to use a smaller set of 'design control points' which only approximate the true geometry. When these points are moved during the optimisation process, they produce a displacement field which can be mapped to the original geometry, generating a new design. This concept is illustrated in Figure 1(a) for a simple 2D aerofoil geometry, where 2 control points are displaced (highlighted with arrows). Compared to directly using the control points required to represent the geometry, this approach drastically reduces the number of design variables whilst maintaining a very large design space.

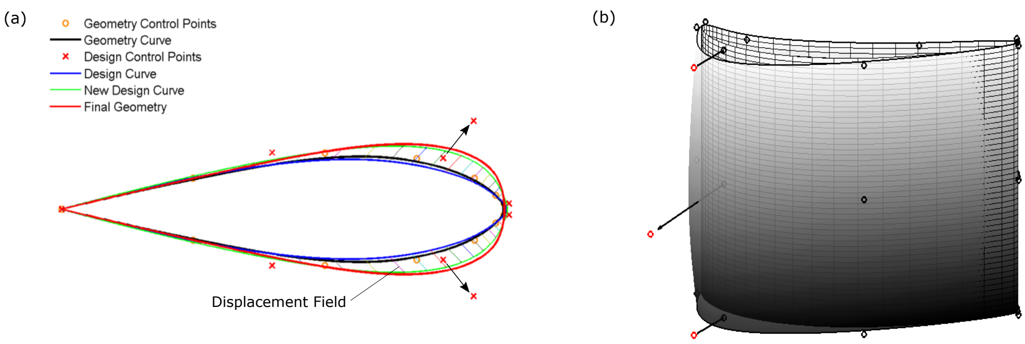


Figure 1: (a) 2D illustration of B-Spline parameterisation method (b) Perturbing the TU Berlin stator blade design by moving 3 control points (red)

Figure 1(b) shows the TU Berlin stator blade approximation (solid grey) and a new design which is achieved by displacing 3 control points (red). The approximate 'design' geometry is represented by 3 radial sections at the hub, tip and mid-span. Each section has 10 control points which gives 30 control points for the whole blade. Each control point is free to move in two directions but due to constraints such as keeping a constant chord, the number of design variables for each section can be reduced to 10. This results in a total of 30 design variables for the optimisation.

The optimisation is performed making use of the Rolls-Royce SOPHY system.[2] Using the given boundary and flow conditions, HYDRA[3] is used for the CFD solver. The optimisation technique used in this study was the trust based, Meta-Assembly Method (MAM) from the SOFT[2] library. This reduces the number of computations necessary for large scale optimisations by using localised design of experiments (DoE) that can be dynamically moved. The results from a generation of simulations (chosen by DoE) are taken and the sub-optimal point then calculated. The search then moves to near this point, where a new generation is created and the process repeated until an optimum design is found.

To assess the efficacy of the B-spline parameterisation method, a comparison is made against a Free-Form-Deformation (FFD) optimisation. The advantages and pitfalls of the approach will be thoroughly discussed during the presentation.

References

[1] John, A., Shahpar, S., Qin, N., *'Alleviation of Shock-Wave Effects on a Highly Loaded Axial Compressor Through Novel Blade Shaping'*, ASME turbo Expo, Seoul, South Korea, GT2016-57550, 2016.

[2] Shahpar, S., *'SOPHY: An integrated CFD based Automatic Design Optimisation System'*,ISABE-2005-1086, 2005.

[3] Lapworth, L., *'Hydra CFD: A Framework for Collaborative CFD Development'* , International Conference on Scientific and Engineering Computation, Singapore, 2004.

Multidisciplinary Design Optimization of Aero-Engine Fan Blades

Christopher Chahine *
Von Karman Institute for Fluid Dynamics, Rhode-Saint-Genese, 1640, Belgium

Tom Verstraete †
Queen Mary University of London, London, E1 4NS,United Kingdom

Li He‡
University of Oxford, Oxford, OX2 0ES, United Kingdom

In this paper we present the application of a multidisciplinary and multiobjective optimization system to the design of a transonic fan blade for high bypass ratio turbofan engines. The optimization includes both aerodynamic and structural performance criteria and is based on a two-level strategy consisting of a Differential Evolution algorithm coupled to a Kriging metamodel in order to speed up the optimization process. High-fidelity performance evaluations are carried out by means of 3D Computational Fluid Dynamics and Computational Structural Mechanics analysis tools. The optimization process involves the evaluation of multiple key operating points of the aircraft mission, including top-of-climb, cruise and take-off performances.

I. Introduction

In this paper the application of a multidisciplinary and multiobjective optimization system to the design of a high bypass ratio aero-engine fan blade is presented. The optimization method enables the concurrent evaluation of aerodynamic and structural performance criteria, therefore facilitating the identification of the interaction of disciplines and allowing the design to progress towards global optimal solutions in a reduced design time.

II. Optimization system

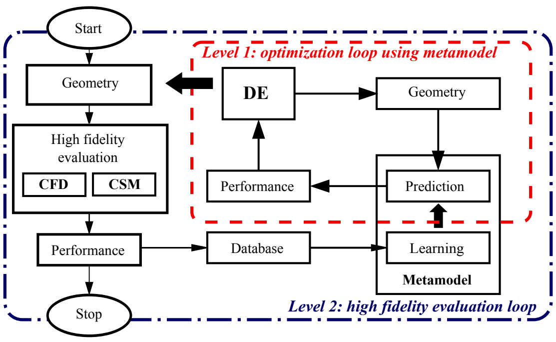


Figure 1. Optimization system.

*DPhil Candidate, Turbomachinery and Propulsion Department, Chaussee de Waterloo 72, 1640 Rhode-Saint-Genese, Belgium and Department of Engineering Science, University of Oxford, Osney Mead, Oxford, OX2 0ES, United Kingdom.
†Visiting Professor, School of Engineering and Material Science, E1 4NS Mile End Road, London, United Kingdom.
‡Professor, Chair of Computational Aerothermal Engineering, Department of Engineering Science, University of Oxford, Osney Mead, Oxford, OX2 0ES, United Kingdom.

A flow chart of the optimization system used in present work is shown in Fig. 1.[1,2] The system is based on a two-level approach with a Kriging metamodel being applied as a continuously updated replacement of the computationally expensive high fidelity evaluation tools. On the first level a multi-objective Differential Evolution algorithm is used to optimize the designs solely based on the Kriging prediction. In order to assess and improve the accuracy of the Kriging metamodel, the best performing samples are automatically re-evaluated by the high-fidelity evaluation chain consisting of CFD and CSM analysis codes on the second level.

III. Fan blade parametrization

The geometry of the fan blade is defined by parametric Bézier and B-Spline curves which specify the blade chord, blade angles, the thickness distributions at hub and tip sections and the profile stacking axis by lean and sweep. The blade metal angles at the leading edge, trailing edge and an intermediate point as well as the chord length are defined by spanwise B-Spline curves. Control points for these distributions are defined on four spanwise positions which are being fixed for three of the points at 0, 50 and 100% span. The blade thicknesses at hub and tip sections are defined by B-Spline curves. Both distributions can be scaled independently by a uniform scaling factor, therefore allowing thickness changes without altering the actual distributions. In addition, the number of blades is allowed to be modified resulting in a total of 26 optimization parameters.

IV. Results

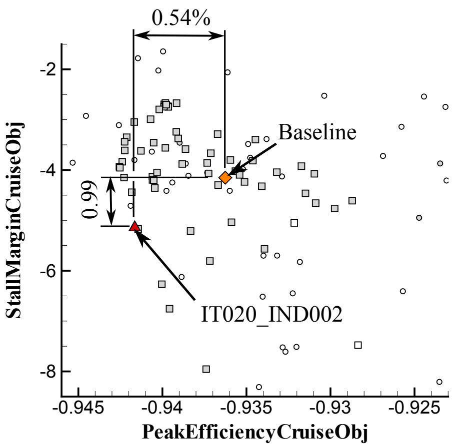


Figure 2. Objective space after 20 iterations. Circles indicate DOE samples while squares indicate designs generated during the optimization. Deigns satisfying the constraints are shown with grey filling.

An overview of the objective space after 20 optimization iterations (290 high-fidelity evaluations) is shown in Fig. 2. For visualization purposes the plot in Fig. 2 was scaled to show the set of feasible designs which are shaded in gray, therefore not all DOE samples are visible. For the chosen design from the Pareto front an efficiency improvement of 0.54 % and a stall margin improvement of 0.99 were obtained with respect to the baseline design.

References

[1] Storn, R., Price, K., *Differential Evolution - A simple and efficient adaptive scheme for global optimization over continuous spaces*, Journal of Global Optimization 11 (4), 1997, pp. 341-359.

[2] Verstraete, T., *CADO: a computer aided design and optimization tool for turbomachinery applications*, in: 2nd International Conference on Engineering Optimization, Lisbon, Portugal, 2010.

Generating Gradients for Turbomachinery Applications using the Discrete Adjoint Method

Georgios Ntanakas*†and Marcus Meyer*

*Rolls-Royce Deutschland, Blankefelde-Mahlow, 15825, Germany

†School of Mechanical Engineering, National Technical University of Athens, Athens, 15780, Greece

The steady adjoint method is already efficiently and actively employed for gradient-based CFD optimization problems in many industrial applications.[3,8,10,11,13] In almost all of those examples, the steady state assumption is used in order to reduce the computational cost of the optimization. However, the recent increase in the availability of compute power is currently leading to a paradigm shift: The use of unsteady simulations in time scales compatible with industrial design cycles is now becoming feasible.

Especially in turbomachinery applications, where the flow is inherently unsteady due to the interaction of stationary and rotating blade rows, unsteady simulations are necessary to avoid the simplifying assumption of the so-called mixing plane coupling[14,15] between blade rows. Instead, a sliding plane coupling is used to increase the model accuracy.[1,2]

In order to enable efficient gradient-based optimization of these unsteady models, it is thus necessary to develop an unsteady adjoint capability: The URANS flow equations are solved using a dual time-stepping technique until a periodic solution is obtained. The objective function of interest is usually defined as the mean of a scalar value of interest (for example the efficiency of the compressor or turbine under consideration) over the period T of the periodic flow solution. The length of the period is defined by the blade count of the blade rows and the rotational speed, from which the dominating frequencies that occur in the model can be calculated. The approach described in this paper can also be applied to non-periodic flows but for the scope of this work and without loss of generality, periodicity of the solution is assumed. Storing the primal flow is essential, since the adjoint solver needs the primal flow during the backwards-in-time marching of the adjoint solver. In the present paper, an overview of the discrete unsteady adjoint method is presented. The final equations are derived and the connection with the steady equations is discussed. Moreover, the additional disk-storage and time cost that is introduced by the unsteadiness is evaluated. The method is applied to testcases to show its capability to enable the efficient optimization of unsteady turbomachinery models.

References

[1]D. Amirante, N. J. Hills, and C. J. Barnes. A moving mesh algorithm for aero-thermo-mechanical modelling in turbomachinery. *International Journal for Numerical Methods in Fluids*, (October 2007):601–629, 2008.

[2]M. Burgos, J. Contreras, and R. Corral. Efficient Edge-Based Rotor/Stator Interaction Method. *AIAA Journal*, 49(1):19–31, jan 2011.

[3]C. Frey, H.-P. Kersken, and D. Nürnberger. The Discrete Adjoint of a Turbomachinery RANS Solver. *ASME Conference Proceedings*, 2009(48883):345–354, 2009.

[4]M. B. Giles and N.A. Pierce. An Introduction to the Adjoint Approach to Design. pages 393–415, 2001.

[5]M.B. Giles. On the use of Runge-Kutta time-marching and multigrid for the solution of steady adjoint equations. 2000.

[6]L. Hascoët and V. Pascual. The Tapenade Automatic Differentiation tool: Principles, Model, and Specification. *ACM Transactions On Mathematical Software*, 39(3), 2013.

[7]A. Jameson, S. Shankaran, and L. Martinelli. Continuous adjoint method for unstructured grids. *AIAA Journal*, 46(5):1226–1239, May 2008.

[8]N. Kroll, N. R. Gauger, J. Brezillon, R. Dwight, . Fazzolari, D. Vollmer, K. Becker, H. Barnewitz, V. Schulz, and S. Hazra. Flow simulation and shape optimization for aircraft design. *Journal of Computational and Applied Mathematics*, 203(2):397 – 411, 2007.

[9]K. Mani and D.J. Mavriplis. Geometry Optimization in Three-Dimensional Unsteady Flow Problems using the Discrete Adjoint. 2013.

[10]A. C. Marta, S.m Shankaran, and A. Stein. Blade Shape Optimization using a RANS Discrete Adjoint Solver. *2nd International Conference on Engineering Optimization*, pages 1–10, 2010.

[11]M. Meaux and M. Cormery. Viscous aerodynamic shape optimization based on the discrete adjoint state for 3d industrial configurations. m:1–19, 2004.

[12]E. J. Nielsen, B. Diskin, and N. K. Yamaleev. Discrete adjoint-based design optimization of unsteady turbulent flows on dynamic unstructured grids. *AIAA Journal*, 48(6):1195–1206, Jun 2010.

[13]E. M. Papoutsis-Kiachagias and K. C. Giannakoglou. Continuous adjoint methods for turbulent flows, applied to shape and topology optimization: Industrial applications. *Archives of Computational Methods in Engineering*, pages 1–45, 2014.

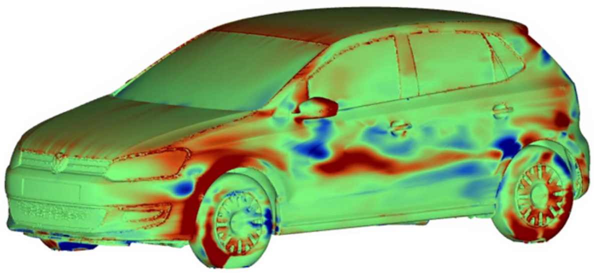
[14]S. Rodrigues and A. Marta. Discrete adjoint mixing-plane formulation for multi-stage turbomachinery design. *Congresso de Metodos Numericos em Engenharia*, 2015.

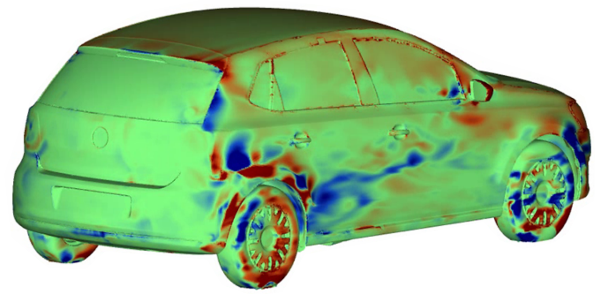
[15]D. X. Wang and L. He. Adjoint Aerodynamic Design Optimization for Blades in Multistage Turbomachines-Part I: Methodology and Verification. *Journal of Turbomachinery*, 132(2):021011, 2010.

Acknowledgements

This work was supported by the *AboutFlow* project (`http://aboutflow.sems.qmul.ac.uk`), funded by the European Commission under FP7-PEOPLE-2012- ITN-317006.

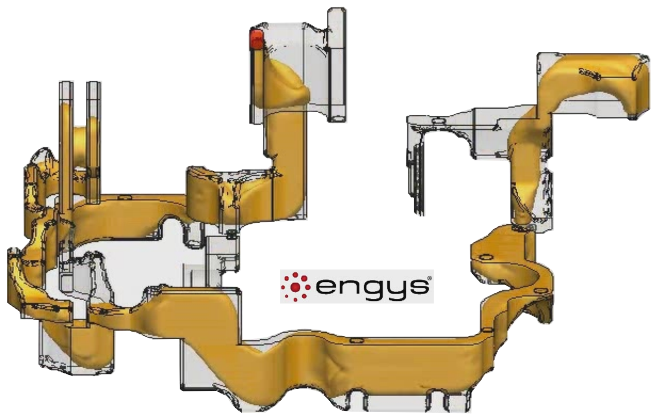
Index of authors

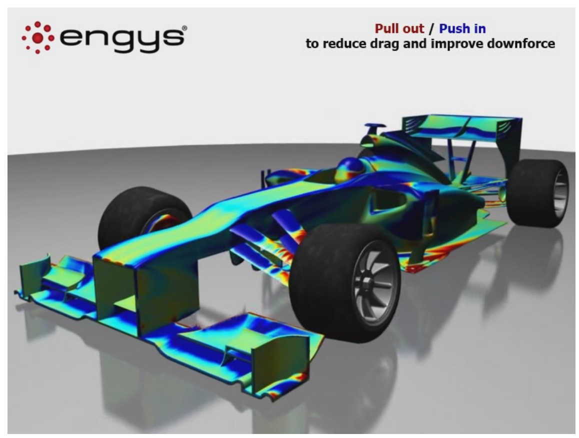


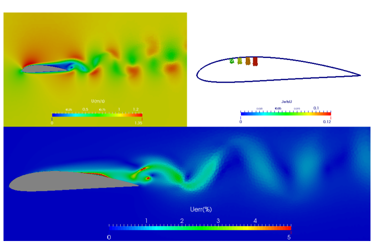


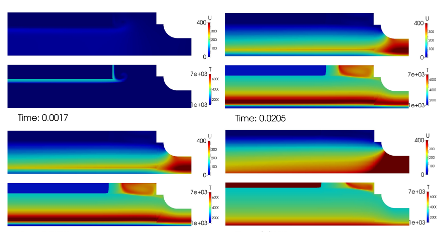
International Conference on Numerical Optimisation Methods for Engineering Design

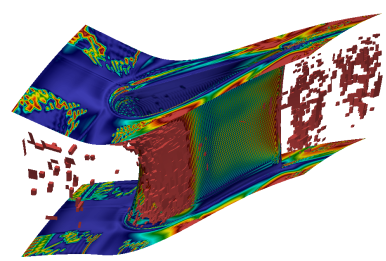
18-20 July 2016

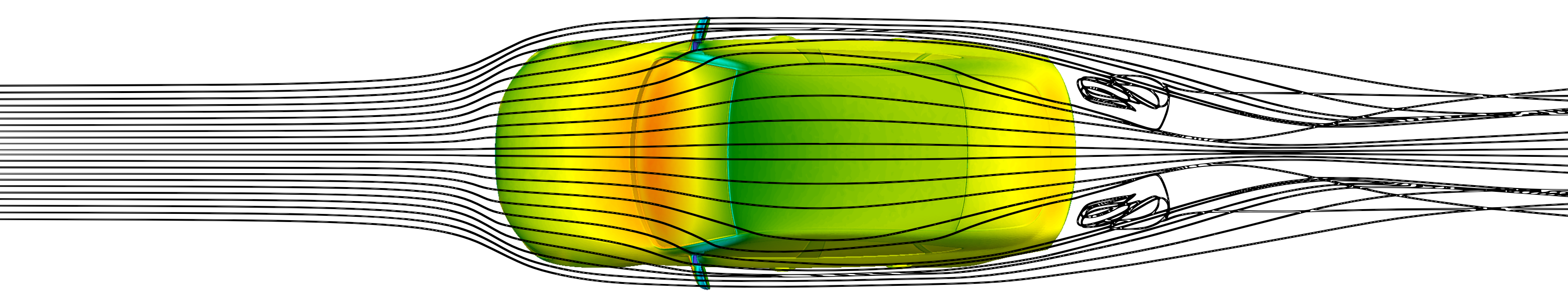












Images supplied by the partners of the Amedeo, About Flow, IODA and Maddog projects funded by the EC.