fluids

MDPI

Editorial

Flow-Based Optimization of Products or Devices

Nils T. Basse

Elsas väg 23, 423 38 Torslanda, Sweden; nils.basse@npb.dk

Received: 14 April 2020; Accepted: 17 April 2020; Published: 22 April 2020

Keywords: flow-based optimization; internal and/or external flow; modelling and simulation; computational fluid dynamics; measurements and theory

---

1. Introduction

Flow-based optimization of products and devices is an immature field compared to corresponding topology optimization based on solid mechanics. However, it is an essential part of component development with both internal and/or external flow.

Flow-based optimization can be achieved by e.g., coupling of computational fluid dynamics (CFD) and optimization software; both open-source and commercial options exist. The motivation for flow-based optimization can be to improve performance, reduce size/cost, extract additional information or a combination of these objectives. The outcome of the optimization process may be geometries which are more suitable for additive manufacturing (AM) instead of traditional subtractive manufacturing.

This MDPI Fluids Special Issue (SI) is a two-fold effort to:

- Provide state-of-the-art examples of flow-based optimization; Table 1 contains an overview of the topics treated in this SI. Also included are the various Quantities of Interest (QoI).
- Present "A Review of Topology Optimisation for Fluid-Based Problems" by Alexandersen and Andreasen [1].

Table 1. Overview of Special Issue research contributions: Applications and Quantities of Interest.

Paper	Application	Quantities of Interest
Kumar et al. [2]	Wind turbine	Power and torque coefficients
Rogié et al. [3]	Microchannel evaporator	Heat transfer and pressure drop
Alexias et al. [4]	Longer static mixing device	Mixture uniformity and pressure drop
Alexias et al. [4]	Shorter static mixing device	Mixture uniformity and pressure drop
Olivetti et al. [5]	Valve	Mass flow rate
Parker et al. [6]	Accelerated wind bodies	Pressure coefficient and velocity
Grossberg et al. [7]	Dispersed multiphase flow	Mass flow rate
Guerrero et al. [8]	Cylinder	Surface area
Guerrero et al. [8]	Static mixer	Velocity distribution
Guerrero et al. [8]	Ahmed bodies	Normalized drag coefficient

2. Research

The research papers are briefly introduced in chronological order; methods and tools applied are summarized in Table 2. Note that all CFD simulations are steady-state and that all simulation-based methods include CAD-based operations to some extent.

Table 2. Overview of the Special Issue research contributions: Optimization methods and tools. Abbreviations: Computational Fluid Dynamics (CFD), Design of Experiments (DoE), Design Space Exploration (DES) and Design Optimization (DO).

Paper	Methods	Tools
Kumar et al. [2]	Parametric optimization	2D CFD: Turbulent flow
Rogié et al. [3]	Parametric optimization	3D CFD: Turbulent flow
Alexias et al. [4]	Continuous adjoint	3D CFD: Laminar flow
Olivetti et al. [5]	Automated DoE	Optimization tool and 3D CFD: Turbulent flow
Parker et al. [6]	Smooth and corrugated cylinder	Measurements of pressure and velocity
Grossberg et al. [7]	Continuous adjoint	Derivation of the adjoint drift flux equations
Guerrero et al. [8]	Cloud-based DSE and DO	Optimization tool and 3D CFD: Turbulent flow

The paper by Kumar et al. [2] is on the topic of small-scale decentralized wind power generation; the authors propose an adaptive hybrid Darrieus turbine (AHDT) to overcome issues experienced by Savonius and Darrieus wind turbines. The AHDT has a Savonius rotor nested inside a Darrieus rotor, where the Savonius rotor can change shape. Optimization consists of changing the diameter of the Savonius rotor while keeping the Darrieus rotor diameter fixed. 2D CFD simulations using the $k-\omega$ shear-stress transport (SST) turbulence model are carried out to study the hybrid turbine performance. The torque coefficient is optimized, which is defined as the ratio of generated aerodynamic torque to the available torque in the wind. The corresponding power coefficient for different tip speed ratios is also characterized. Flow interaction between the Savonius rotor in closed configuration and the Darrieus rotor blades takes place due to the formation of Kármán vortices.

Rogié et al. [3] compare new microchannel evaporator designs to a baseline finned-tube evaporator; the new designs have drainage slits for improved moisture removal with triangular shaped plain fins. Optimization is carried out by varying the geometry (transverse tube pitch and triangular fin pitch) and the inlet velocity while keeping a constant wall temperature of tube and fin. 3D $k-\omega$ SST CFD simulations were done to establish heat transfer coefficients and pressure drop, both as a function of tube rows. These results were in turn used to develop Colburn j-factor and Fanning f-factor correlations. It was found that the entrance region is very important for heat transfer and that the new designs transfer more heat per unit volume than the baseline.

The continuous adjoint method is applied by Alexias and Giannakoglou [4] to study two-fluid mixing devices. The authors consider laminar flow of two miscible fluids and change baffle shapes and angles to optimize (i) mixture uniformity at the exit and (ii) the total pressure loss occurring between the inlets and the outlet. These two objectives are used to construct a single target function. The primal (flow) and adjoint field equations are solved and thereafter the sensitivity derivatives are found. Two mixing devices are treated, one longer (with 7 baffles) and one shorter (with 4 baffles). Both have two inlets and one outlet. Three optimization scenarios are tested using combinations of node-based parametrization (NBP) and positional angle parametrization (PAP). Results are presented and it is demonstrated that the shorter mixing device has a lower pressure drop but also worse mixing quality than the longer mixing device.

Olivetti et al. [5] optimize a four-way hydropiloted valve by combining an optimization tool (with integrated parametric geometry) and CFD simulations. The 3D CFD simulations uses the standard $k-\varepsilon$ turbulence model. The shape of two ports of the valve are optimized to maximise mass flow rate for a fixed static pressure difference between the two ports. A Design of Experiments (DoE) sequence is generated with a Sobol algorithm which determined that 8 design variables resulting in 90 variants should be simulated. The Sobol sequence resulted in a significant increase of the mass flow rate. A second optimization step was done on the best Sobol sequence design using a 2-level tangent search (Tsearch) method which led to a further improvement. Experiments confirmed the findings obtained using the CFD-based optimization.

Cylindrical bodies for “accelerated wind” applications are experimentally characterized by Parker and Bohl [6]. Here, one aims to enhance power extraction from wind by adding a structure near the rotor to increase the flow velocity, i.e., to increase the kinetic energy of the wind before it reaches the wind turbine blades. Two short aspect ratio cylindrical bodies are tested, a corrugated and a smooth cylinder. The cylindrical bodies are tested in a wind tunnel using varying Reynolds number (*Re*); pressure taps are placed in the bodies and the velocity is measured with hot-wire probes. End effects are found to be important. Both bodies demonstrated increased flow speed, but gauged by the pressure coefficient and velocity, the smooth cylinder exhibited better performance than the corrugated cylinder.

The continuous adjoint method is applied to dispersed multiphase systems by Grossberg et al. [7]. A drift-flux model is studied, where the two separate phases are considered as a single mixture phase. This is a simplification compared to the two-fluid formulation. The transport of the dispersed phase is modelled using a drift equation; this equation, along with mixture-momentum and mixture-continuity equations, forms the drift flux (primal) equations. The adjoint drift flux equations with a Darcy porosity term are derived under the frozen turbulence (or constant mixture turbulent viscosity) assumption. The corresponding boundary conditions for the adjoint variables are also calculated. Application examples are documented for wall-bounded flows, where (i) adjoint boundary conditions, (ii) the objective function and (iii) the settling (drift) velocity are derived. The objective function is the mass flow rate of the dispersed phase at the outlet.

Guerrero et al. [8] present an engineering design framework with a cloud-based parametrical CAD application which can be used on any platform without the need for a local installation. The optimization loop is fault-tolerant and scalable in the sense that both concurrent and parallel simulations can be deployed. Two methods are used for optimization: Design Space Exploration (DSE) and Design Optimization (DO). DO converges to an optimal design, either using a (i) gradient-based or (ii) derivative-free method. In contrast, DSE is used to explore the design space in a methodical fashion without converging to an optimum. Results from DSE provide more information to the engineer than DO and can also be used for e.g., surrogate-based optimization studies which are orders of magnitude faster than working at the high fidelity level. A useful approach can be to carry out a DSE as a first step, followed by a DO. Three numerical experiments are documented in the paper: The first example minimizes the total surface area of a cylinder with a given volume and serves to introduce the optimization framework. The second example on a static mixer uses 3D CFD simulations with the $k-\varepsilon$ turbulence model and compares velocity profile images using the Structural Similarity Index (SSIM) method. The third example is on changing the inter-vehicle spacing between two Ahmed bodies to calculate the resulting normalized drag coefficient. 3D CFD using the $k-\omega$ SST turbulence model is used and the simulations are compared to measurements.

3. Review

Alexandersen and Andreasen [1] have written the first complete review on topology optimization for fluid-based problems. This research field was started in 2003; at that point in time, topology optimization of solid mechanics had already been an active research area for 15 years. 186 papers are covered by the literature review according to the selection criterion that at least one governing equation for fluid flow must be solved; the topics are summarized in Table 3.

The quantitative analysis of the literature discusses the total number of publications per year and how these are distributed in terms of:

- Design representations, e.g., density-based and level set methods
- Discretization methods, e.g., the finite element method and the lattice Boltzmann method
- Problem types, e.g., pure fluid and conjugate heat transfer
- Flow types, e.g., steady-state and transient laminar flow
- Dimensionality, i.e., 2D or 3D

Recommendations are given, ranging from methods used, to which types of physical problems the community should focus on in the future. Topics covered by the recommendations include:

- Optimization methods
- Density-based approaches
- Level set-based approaches
- Steady-state laminar incompressible flow
- Benchmarking
- Time-dependent problems
- Turbulent flow
- Compressible flow
- Fluid-structure interaction
- 3D problems
- Simplified models or approximations
- Numerical verification
- Experimental validation

Finally, to quote from the Conclusions of the paper, "The community is encouraged to focus on moving the field to more complicated applications, such as transient, turbulent and compressible flows."

Table 3. Overview of flow topics treated in the review paper.

Main Topic	Subtopic (If Applicable)
Fluid flow	Steady laminar flow Unsteady flow Turbulent flow Non-Newtonian fluids
Species transport	
Conjugate heat transfer	Forced convection Natural convection
Fluid-structure interaction	
Microstructure and porous media	Material microstructures Porous media

4. Conclusions

Examples of flow-based optimization research have been provided in this Special Issue along with a complete review of the research field.

There is a natural connection between flow-based optimization and AM, since geometrical shapes resulting from optimization may be challenging to realize using traditional manufacturing methods. Note that Parker and Bohl [6] used AM to manufacture the corrugated cylinder. We recommend researchers in the field to use AM more extensively in the future to test geometries from simulation studies.

Another area where more synergy can be explored is to combine Design Space Exploration and Machine Learning [9,10] as is also mentioned by Guerrero et al. [8].

A range of physical flow phenomena which are suitable for topology optimization exists, see e.g., the list in the Special Issue Information Section [11]. We look forward to following the research field in the future; surely, this is only the beginning!

Acknowledgments: We would like to thank all the authors who contributed to this Special Issue. We are also grateful to all the anonymous reviewers for their help; without the help of qualified reviewers, it would not have been possible to organize this Special Issue. A personal note of appreciation and gratitude to Sonia Guan, the Managing Editor of Fluids, and the editorial staff at the Fluids Office; without their help and assistance, Fluids could not publish high quality papers in a short period of time.

Conflicts of Interest: The author declares no conflict of interest.

References

1. Alexandersen, J.; Andreasen, C.S. A Review of Topology Optimisation for Fluid-Based Problems. *Fluids* **2020**, *5*, 29. [CrossRef]
2. Kumar, P.M.; Surya, M.R.; Sivalingam, K.; Lim, T.-C.; Ramakrishna, S.; Wei, H. Computational Optimization of Adaptive Hybrid Darrieus Turbine: Part 1. *Fluids* **2019**, *4*, 90. [CrossRef]
3. Rogié, B.; Markussen, W.B.; Walther, J.H.; Kærn, M.R. Numerical Investigation of Air-Side Heat Transfer and Pressure Drop Characteristics of a New Triangular Finned Microchannel Evaporator with Water Drainage Slits. *Fluids* **2019**, *4*, 205. [CrossRef]
4. Alexias, P.; Giannakoglou, K.C. Shape Optimization of a Two-Fluid Mixing Device Using Continuous Adjoint. *Fluids* **2020**, *5*, 11. [CrossRef]
5. Olivetti, M.; Monterosso, F.G.; Marinaro, G.; Frosina, E.; Mazzei, P. Valve Geometry and Flow Optimization through an Automated DOE Approach. *Fluids* **2020**, *5*, 17. [CrossRef]
6. Parker, M.; Bohl, D. Experimental Investigation of Finite Aspect Ratio Cylindrical Bodies for Accelerated Wind Applications. *Fluids* **2020**, *5*, 25. [CrossRef]
7. Grossberg, S.; Jarman, D.S.; Tabor, G.R. Derivation of the Adjoint Drift Flux Equations for Multiphase Flow. *Fluids* **2020**, *5*, 31. [CrossRef]
8. Guerrero, J.; Mantelli, L.; Naqvi, S.B. Cloud-Based CAD Parametrization for Design Space Exploration and Design Optimization in Numerical Simulations. *Fluids* **2020**, *5*, 36. [CrossRef]
9. MDPI Fluids Special Issue on "Numerical Fluid Flow Simulation Using Artificial Intelligence and Machine Learning". Available online: https://www.mdpi.com/journal/fluids/special_issues/Artificial_Intelligence_and_Machine_Learning (accessed on 21 April 2020).
10. Brunton, S.L.; Noack, B.R.; Koumoutsakos, P. Machine Learning for Fluid Mechanics. *Annu. Rev. Fluid Mech.* **2020**, *52*, 477–508. [CrossRef]
11. MDPI Fluids Special Issue on "Flow-Based Optimization of Products or Devices". Available online: https://www.mdpi.com/journal/fluids/special_issues/flow_optimization (accessed on 21 April 2020).