upstreamCFD

Plans for a Technical Committee on Turbulence within the OpenFOAM Governance Structure

Charles Mockett

charles.mockett@upstream-cfd.com

Introduction

Introduction & outline of talk

- Since its first open source release in 2004 by OpenCFD Ltd, OpenFOAM has grown to become the leading open source software for Computational Fluid Dynamics
- As even a casual observer of OpenFOAM's progress knows, a key challenge has been to strike the balance between:
  - Welcoming and integrating community developments, and
  - Satisfying end-user requirements for rigorous quality assurance
- ...whilst protecting the open source nature of the code from commercial interests
- ...and enabling business models based on OpenFOAM to flourish
- My opinion: As long as the code and the community remain fragmented, OpenFOAM will always punch below its weight compared to the commercial codes

- This talk:
  - Describe an initiative for community engagement via "OpenFOAM Governance"
  - Introduce the Technical Committee on Turbulence within this structure

Upstream CFD GmbH

- Founded in Berlin in January 2019
- Team of five co-founders:
  - Charles Mockett (MD), Marian Fuchs, Felix Kramer, Thilo Knacke & Norbert Schönwald
  - Established team with a total of 60 years professional experience
- Areas of expertise:
  - Turbulence modelling
  - Aeroacoustics
  - Numerical methods
  - Optimisation
  - High-performance computing

- Services offered:
  - R&D: Improved CFD/CAA methods
  - Automated & adaptive CFD/CAA processes for specific applications
  - Aerodynamic and aeroacoustic consulting based on high-fidelity simulations
  - HPC system support

upstreamCFD

OpenFOAM governance framework

OpenFOAM Governance

- Purpose and ambition:
  - Bringing the community together in an open, inclusive and co-operative framework
  - Ensuring the project's longevity, free availability and open-source values
- Launched by ESI-OpenCFD and partners at OpenFOAM User Conference in October 2018
  - For more details: https://www.openfoam.com/governance/
- Structure:
  - Steering Committee
    - Representatives from OpenFOAM's main sponsors in Industry, Academia, Release Authorities and Consultant Organisations
  - Technical Committees
    - Covering all key focus areas for OpenFOAM's development
    - Assess state-of-the-art, need for validation, documentation and further development
    - Committee chairs appointed by Steering Committee, 3 year term, bursary to support efforts & overheads
    - Committee membership open to all, bound by a Code of Conduct

Steering Committee members

<table>
<tr><th>Name</th><th>Affiliation</th><th>Responsibility</th></tr>
<tr><td>Fred Mendonça</td><td>OpenCFD Ltd</td><td>Chair</td></tr>
<tr><td>Karen Kettle</td><td>OpenCFD Ltd</td><td>Administrator</td></tr>
<tr><td>Dr Marcus Renner</td><td>Volkswagen</td><td rowspan="4">Representing OEMs & Support Consultants</td></tr>
<tr><td>Dr Paul Eno</td><td>General Motors</td></tr>
<tr><td>Dr Karl Meredith</td><td>FM Global</td></tr>
<tr><td>Dr Rob Lewis OBE</td><td>TotalSim</td></tr>
<tr><td>Dr.-Ing. Holger Marschall</td><td>Technische Universität Darmstadt</td><td>Representing Research with OpenFOAM</td></tr>
<tr><td>Mike Salari</td><td>OpenCFD Ltd</td><td rowspan="3">Representing interests of release authorities</td></tr>
<tr><td>Prof. Hrvoje Jasak</td><td>Wikki Ltd</td></tr>
<tr><td>Christopher St John</td><td>ESI Group</td></tr>
<tr><td>Dr Andrew Heather</td><td>OpenCFD Ltd</td><td>Release & Maintenance</td></tr>
</table>

Current Technical Committees & chairs

Committee	Chair	Institution
Documentation & Tutorials	Jószef Nagy	Johannes Kepler University Linz
Marine Applications	Kevin Maki	University of Michigan
Meshing	Franjo Juretic	Creative Fields
Multiphase	Holger Marschall	TU Darmstadt
Numerics	Hrvoje Jasak	University of Zagreb
Optimisation	Kyriakos Giannakoglou	National Technical University of Athens
Turbulence	Charles Mockett	Upstream CFD GmbH

upstreamCFD

Technical Committee on Turbulence

Technical Committee on Turbulence

- Objective:
  - To establish OpenFOAM as the first-choice code for turbulent flow simulation in research and industry
- Key to achieving this:
  - Community engagement
  - Quality assurance
- Approach to Committee Membership:
  - Aim for around 5 members
  - Balance of academia and industry
  - Mixture of technical backgrounds (e.g. RANS, scale-resolving methods, wall functions, V&V)
  - Members also from outside the OpenFOAM community, as long as there are no conflicts of interest with the purpose to promote OpenFOAM
  - One committee member from OpenCFD (link to code maintenance & release)
  - Initiative applications always welcome

Committee members (so far)

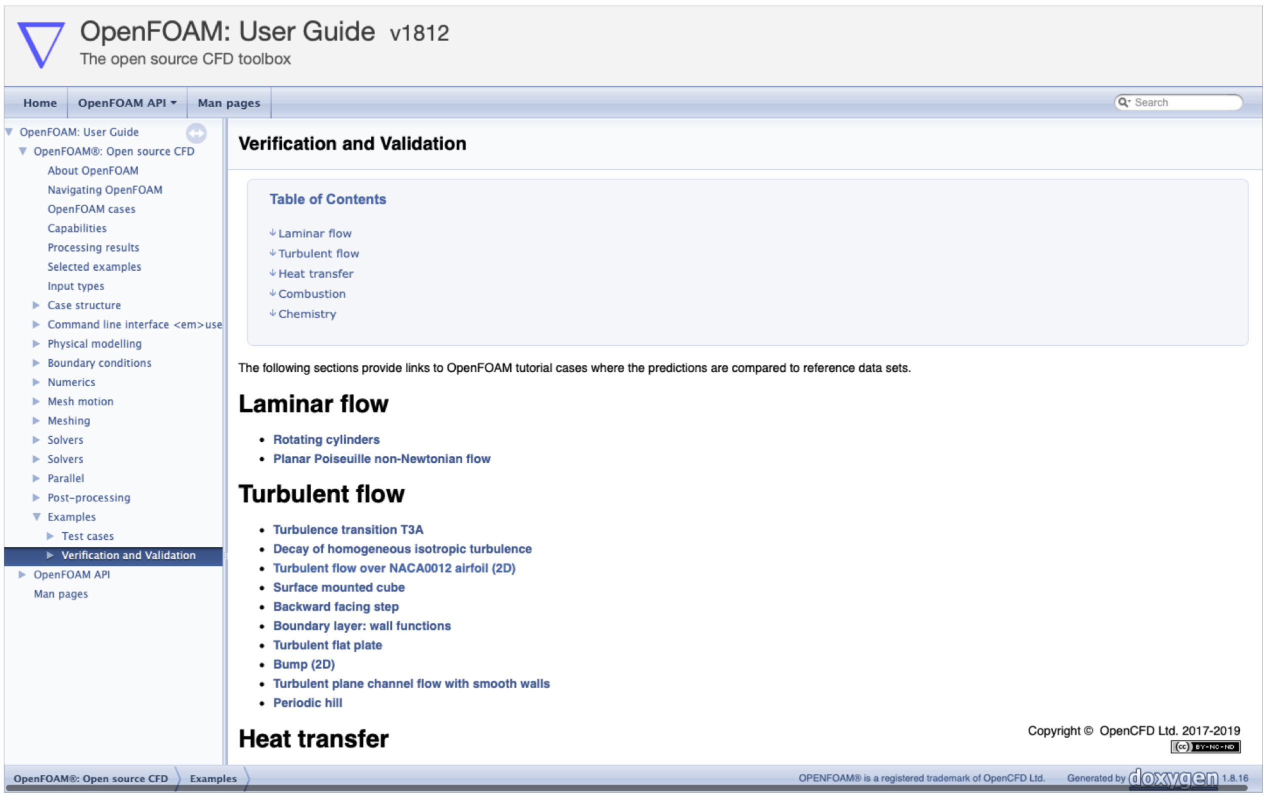
Photo	Institution	Name	Tech. expertise	Application areas
	OpenCFD Ltd	Andy Heather	Verification & Validation Code development	Release & Maintenance
	Upstream CFD GmbH	Dr. Charles Mockett	Hybrid RANS-LES Verification & Validation Numerics for LES	Aerodynamics Aeroacoustics
	Chalmers University of Technology	Dr. Timofey Mukha	Wall-modelled LES LES inlet BCs	Marine
	KTH Royal Institute of Technology	Dr. Stefan Wallin	RANS LES Hybrid RANS-LES	Aeronautics

Planned activities

- Remit: OpenFOAM recommendations to the Steering Committee with respect to turbulence modelling
- Planned activities:
  - Review of current turbulence modelling functionality, validation and documentation
  - Establish links to related Technical Committees (meshing, numerics, …)
  - Gather & publish overview of implementations soon to be available (avoid duplicate work)
  - Promote OpenFOAM's turbulence modelling capabilities for industrial applications
  - Engage with the turbulence modelling research community
    - OpenFOAM as a "Common Assessment Platform"
    - Publish validation test cases and example OpenFOAM results for different models
    - Link to existing databases, e.g. NASA Turbulence Modelling Resource, ERCOFTAC
    - Encourage academics to publish OpenFOAM source code together with papers on new models
  - Provide recommendations w.r.t. turbulence modelling to the Steering Committee upon request
  - Define a road map for turbulence modelling development in OpenFOAM and propose to SC

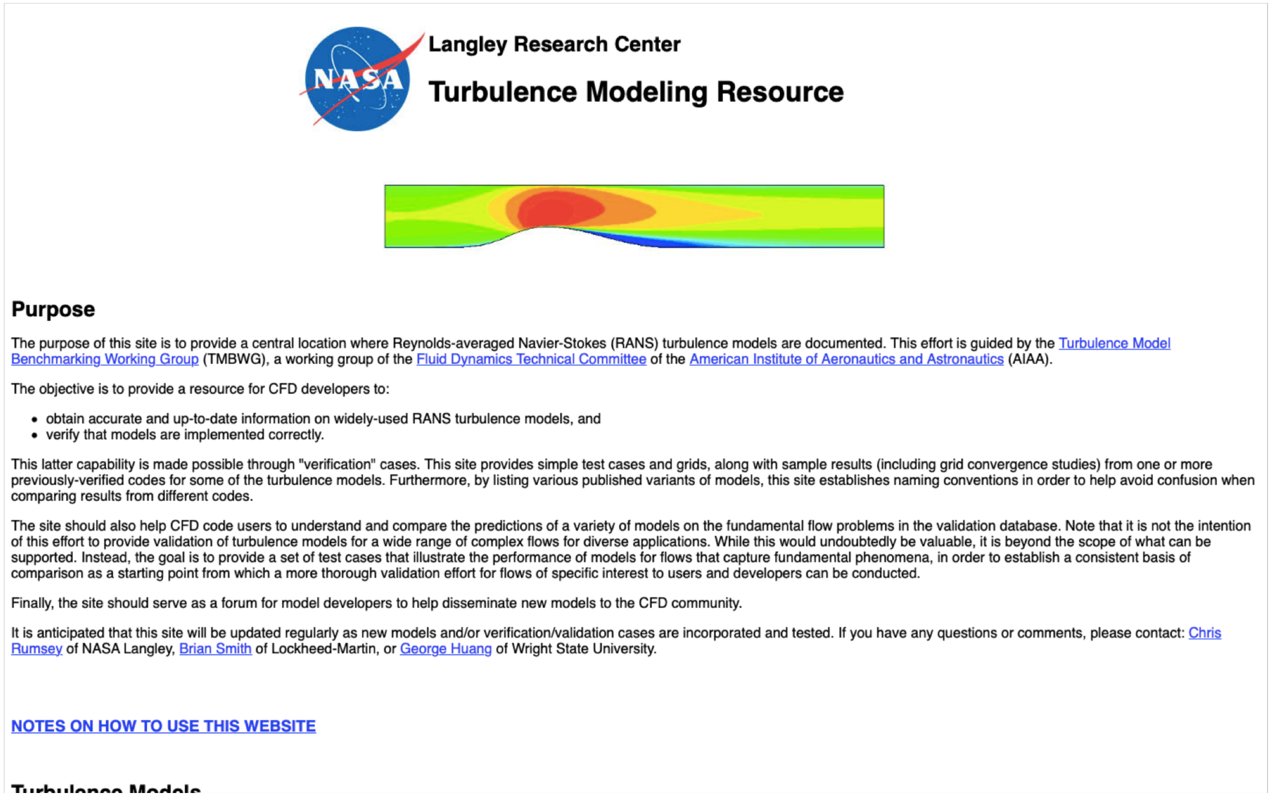
Verification & Validation

- Example* of V&V studies using test cases and data from the NASA Turbulence Modelling Resource**
  - * https://www.openfoam.com/documentation/guides/latest/doc/guide-verification-validation.html
  - ** https://turbmodels.larc.nasa.gov



Verification & Validation

- Example* of V&V studies using test cases and data from the NASA Turbulence Modelling Resource**
  - * https://www.openfoam.com/documentation/guides/latest/doc/guide-verification-validation.html
  - ** https://turbmodels.larc.nasa.gov



Langley Research Center

Turbulence Modeling Resource

Purpose

The purpose of this site is to provide a central location where Reynolds-averaged Navier-Stokes (RANS) turbulence models are documented. This effort is guided by the Turbulence Model Benchmarking Working Group (TMBWG), a working group of the Fluid Dynamics Technical Committee of the American Institute of Aeronautics and Astronautics (AIAA).

The objective is to provide a resource for CFD developers to:

- obtain accurate and up-to-date information on widely-used RANS turbulence models, and
- verify that models are implemented correctly.

This latter capability is made possible through "verification" cases. This site provides simple test cases and grids, along with sample results (including grid convergence studies) from one or more previously-verified codes for some of the turbulence models. Furthermore, by listing various published variants of models, this site establishes naming conventions in order to help avoid confusion when comparing results from different codes.

The site should also help CFD code users to understand and compare the predictions of a variety of models on the fundamental flow problems in the validation database. Note that it is not the intention of this effort to provide validation of turbulence models for a wide range of complex flows for diverse applications. While this would undoubtedly be valuable, it is beyond the scope of what can be supported. Instead, the goal is to provide a set of test cases that illustrate the performance of models for flows that capture fundamental phenomena, in order to establish a consistent basis of comparison as a starting point from which a more thorough validation effort for flows of specific interest to users and developers can be conducted.

Finally, the site should serve as a forum for model developers to help disseminate new models to the CFD community.

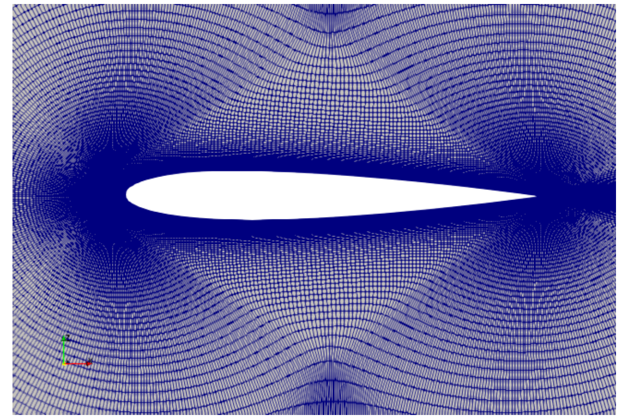
It is anticipated that this site will be updated regularly as new models and/or verification/validation cases are incorporated and tested. If you have any questions or comments, please contact: Chris Rumsey of NASA Langley, Brian Smith of Lockheed-Martin, or George Huang of Wright State University.

NOTES ON HOW TO USE THIS WEBSITE

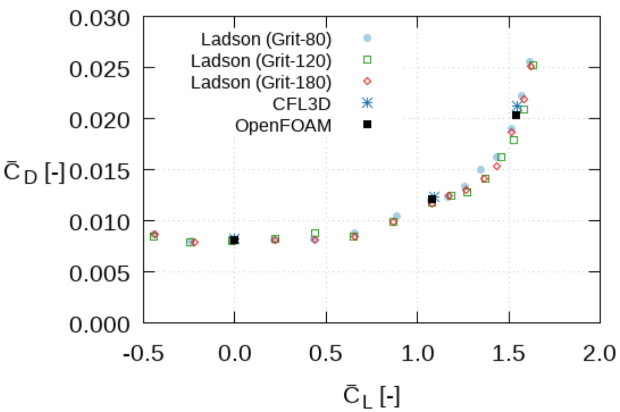
Turbulence Models

Verification & Validation

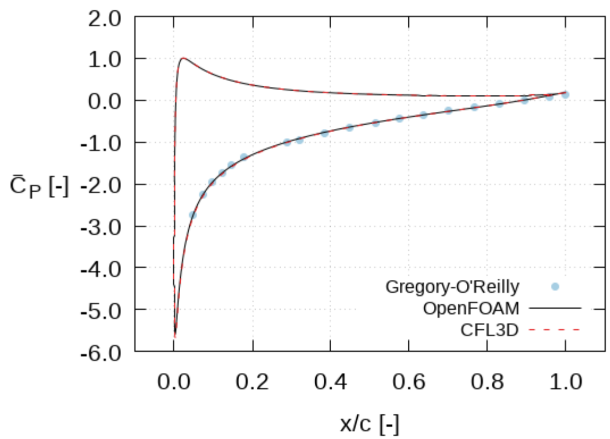
- Example* of V&V studies using test cases and data from the NASA Turbulence Modelling Resource**
  - Turbulent flow over NACA0012 airfoil (2D)
  - $Re_c = 6{\times}10^6$
  - Incompressible, steady-state simulation (*simpleFoam*)
  - Spalart-Allmaras model
  - Fine structured grid (257 x 897)
  - Excellent agreement with measurements and CFL3D code



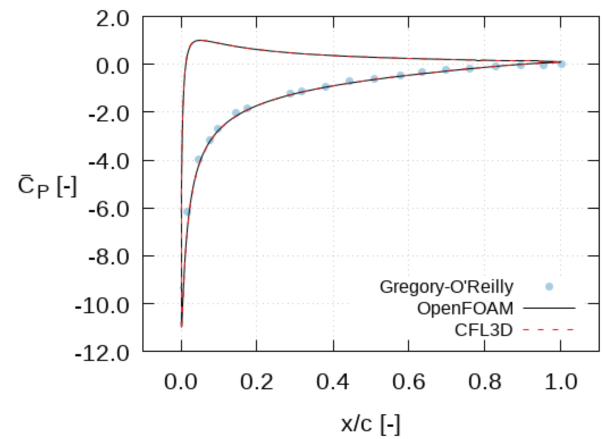
Lift-drag polar



Pressure distribution $(\alpha = 10°)$



Pressure distribution $(\alpha = 15°)$



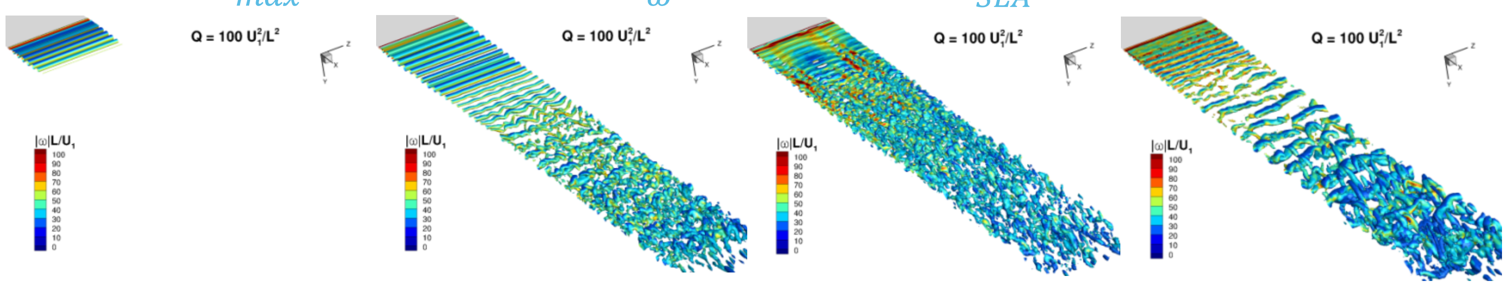
Common Assessment Platform (Go4Hybrid)

upstream CFD

- EU-funded turbulence modelling research project “Go4Hybrid”
  - “Grey-area mitigation for hybrid RANS-LES models”
  - Numerous partners developing different turbulence models in different codes
  - How to draw fair conclusions about model performance?
- “Common Assessment Platform”:
  - A selection of models from the project were implemented in OpenFOAM
  - Direct comparison on the same grid with the same numerics
  - Example: Plane shear layer downstream of splitter plate

Resolved turbulence in the shear layer indicating level of grey area mitigation:

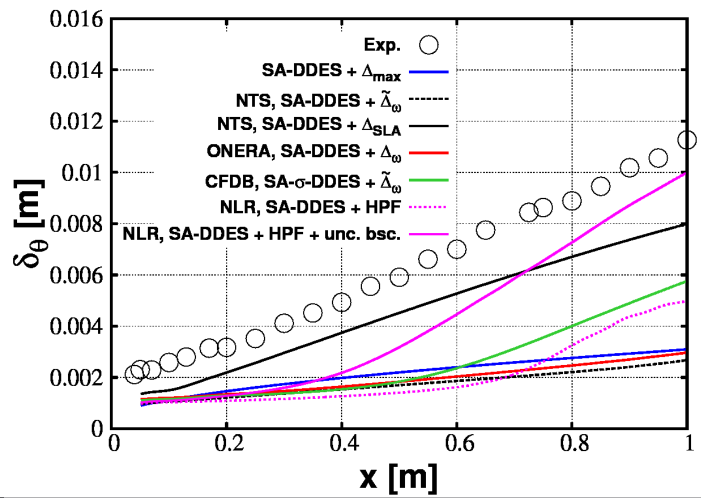
SA-DDES + Δ_{max} | SA-σ-DDES + $\widetilde{\Delta}_{\omega}$ | SA-DDES + Δ_{SLA} | SA-DDES + HPF + BS



Common Assessment Platform (Go4Hybrid)

- EU-funded turbulence modelling research project “Go4Hybrid”
  - “Grey-area mitigation for hybrid RANS-LES models”
  - Numerous partners developing different turbulence models in different codes
  - How to draw fair conclusions about model performance?
- “Common Assessment Platform”:
  - A selection of models from the project were implemented in OpenFOAM
  - Direct comparison on the same grid with the same numerics
  - Example: Plane shear layer downstream of splitter plate

Development of shear layer momentum thickness downstream of splitter plate compared between different grey area mitigation approaches



upstreamCFD

Conclusion & Outlook

Conclusion & Outlook

- The current status and ambitions of a Technical Committee on Turbulence within the OpenFOAM Governance structure have been introduced
- It's early days, but it is hoped that such activities can address a critical remaining weakness of OpenFOAM: Fragmentation of the community
- Successful community engagement has enormous potential to propel OpenFOAM to "best in class" status (not just within the open source category)
  - In terms of expertise, I don't believe that any single entity can compete with an engaged and well-organised community
- Liberal use of Big Words such as "open", "transparent" and "inclusive" has been made
  - If you're sceptical:
    - This is probably a healthy sign!
    - Please consult the available information*, challenge us with difficult questions and put it to the test

- If you're interested in getting involved, please contact* the chair of the relevant Technical Committee, or the Steering Committee directly

- * https://www.openfoam.com/governance/

upstreamCFD

Thank you for your attention