POLITECNICO
MILANO 1863

RED Fluid Dynamics

From medical devices to surfboards design: CFD consulting with OpenFOAM

Riccardo Rossi, PhD
Head and Founder

1st Italian OpenFOAM User Meeting
Politecnico di Milano, October 19th, 2022

RED Fluid Dynamics

About RED

History

- Established in **2016**
- **50+ projects** and **30+ collaborations** up to date
- 30+ years experience in **CFD**

Areas of Expertise

- Research and development on **unstructured finite-volume schemes**
- From **high-fidelity simulations** (DNS, LES) to **RANS modeling** of turbulent flows
- Advanced **modeling of turbulent transport**

Applications Experience

- Automotive and transportation
- Architecture and construction
- Energy and environment
- **Healthcare and medical devices**
- Hydraulics and marine
- Process engineering
- **Sports engineering**

RED Team

Riccardo Rossi, Head and Founder

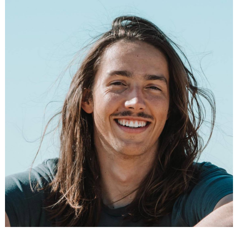
MSc in Mechanical Engineering, PhD in **Thermo-Fluid Dynamics**
20+ in CFD, former **University of Bologna** and **Stanford University**

Enrico Bezzi, Senior CFD Analyst

BSc in **Mechanical Engineering**
5+ in CFD, specialist in **automotive applications**

Federico Angius, Junior CFD Analyst

MSc in **Civil Engineering**
4+ in CFD, specialist in **boardsports applications**



Alfred Reid, Junior CFD Analyst

MSc in **Computational Engineering**
2+ in CFD, specialist in **pharmaceuticals applications**

Andrea Ruju, Wave Modeling Specialist

MSc Environmental Engineering, PhD in **Coastal Engineering**
15+ in wave modeling, former IH **Cantabria Plymouth University**

CFD tools (1/2) – OpenFOAM®

Generalities

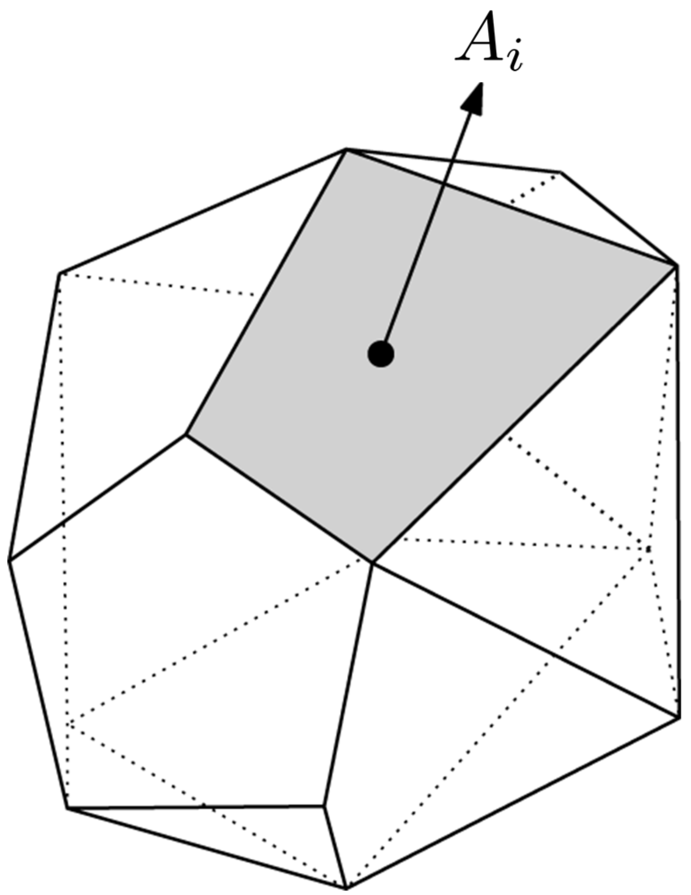
- OpenFOAM® stands for **Open Source** Field Operation and Manipulation
- OpenFOAM® is first and foremost a **C++ library** used to solve partial differential equations (PDEs), and ordinary differential equations (ODEs)
- It is licensed under the GNU **General Public License** (GPL)

Numerics

- **Finite-Volume Method** (FVM) based solver
- Collocated polyhedral unstructured meshes
- **Second order accuracy** in space and time
- MPI-based **parallel execution**

Pros and cons

- **License free**, multiphysics, highly customizable
- Steep learning curve, high expertise, **no GUI**



Generic polyhedral cell volume

CFD tools (2/2) – Cineca

Galileo 100 characteristics:

- **Model:** DUal-Socket Dell PowerEdge
- **Architecture:** Linux Infiniband Cluster
- **Nodes**: 554 (+10 login nodes)
- **Processors:** 2xCPU x86 Intel Xeon Platinum 8276-8276L (24c, 2.4Ghz)
- **Cores:** 48 cores/node
- **Accelerators:** 2xGPU nVidia V100 PCIe3 with 32GB RAM on 36 Viz nodes
- **RAM**: 384GB (+ 3.0TB Optane on 180 fat nodes)
- **Internal Network:** Mellanox Infiniband 100GbE
- **Peak performance single node:** 3.53 TFlop/s
- **Peak performance - total :** about 2 PFlop/s
- Remote access via **visualization services**
- **Start of production:** October 2021

IBM-type cluster installation at Cineca. (source: Cineca website)

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- 2014: teaching CFD using OpenFOAM at Stanford University
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- 2014: teaching CFD using OpenFOAM at Stanford University
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

RED Fluid Dynamics

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

RED Fluid Dynamics

1996: **(no CFD experience)**

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: **(8+ years OpenFOAM experience, 5 OpenFOAM training offered)**

OpenFOAM in RED

1996: (no CFD experience)

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

RED Fluid Dynamics

1996: **(no CFD experience)**

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: (8+ years OpenFOAM experience, 5 OpenFOAM training offered)

OpenFOAM in RED

1996: **(no CFD experience)**

- 1997: first CFD simulation using Fluent V4
- 2002: CTFD-Lab founded at the University of Bologna
- 2005: first ever external CFD of a Ducati bike using Fluent
- **2014: teaching CFD using OpenFOAM at Stanford University**
- 2015: first ever external CFD of the Pagani Huayra using OpenFOAM
- 2016: RED founded
- 2019: first OpenFOAM training for Chiesi Farmaceutici

Today: **(8+ years OpenFOAM experience, 5 OpenFOAM training offered)**

RED Fluid Dynamics

From medical devices...

Overview

In 2018, we became **partner of Chiesi Farmaceutici S.p.A.** to provide technical support and training in the use of OpenFOAM for the simulation of **medical devices** and **drug delivery** in the human body as well as for equipment and machinery for **drug production**.

Why OpenFOAM:

“Working with open-source software, like OpenFOAM, allows for accessing **high-quality tools free of charge** as well as for **better integration/optimization with computing platforms** and, above all, to **keep the tools available to research groups** which invested in their development”

– Andrea Benassi, Global Technical Development Digital, Data & Modeling Dept.

Pressurized Metered Dose Inhaler for asthma treatment.
(courtesy of Chiesi Farmaceutici S.p.A.)

Modeling flashing in a pMDI (1/2)

In a **pressurized Metered Dose Inhaler** or pMDI, **flash-boiling** of the liquid propellant stored in the metallic canister forms a **gas/liquid mixture** which, flowing through a nozzle, forms the aerosol emitted from the mouthpiece.

Challenges using OpenFOAM:

- **No compressible solver** with phase-change (was) available
- Standard phase-change models (strictly) valid only for **homogeneous conditions**
- **No heat transfer** and **no variable saturation pressure** available in the standard solver

Pressurized Metered Dose Inhaler for asthma treatment.
(courtesy of Chiesi Farmaceutici S.p.A.)

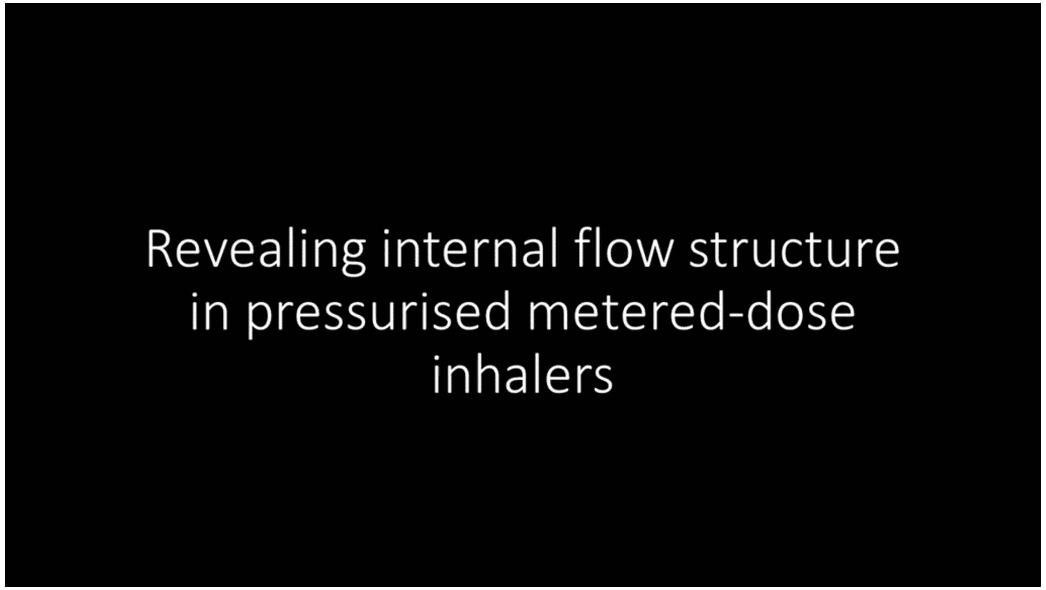
RED Fluid Dynamics

Modeling flashing in a pMDI (1/2)

In a **pressurized Metered Dose Inhaler** or pMDI, **flash-boiling** of the liquid propellant stored in the metallic canister forms a **gas/liquid mixture** which, flowing through a nozzle, forms the aerosol emitted from the mouthpiece.

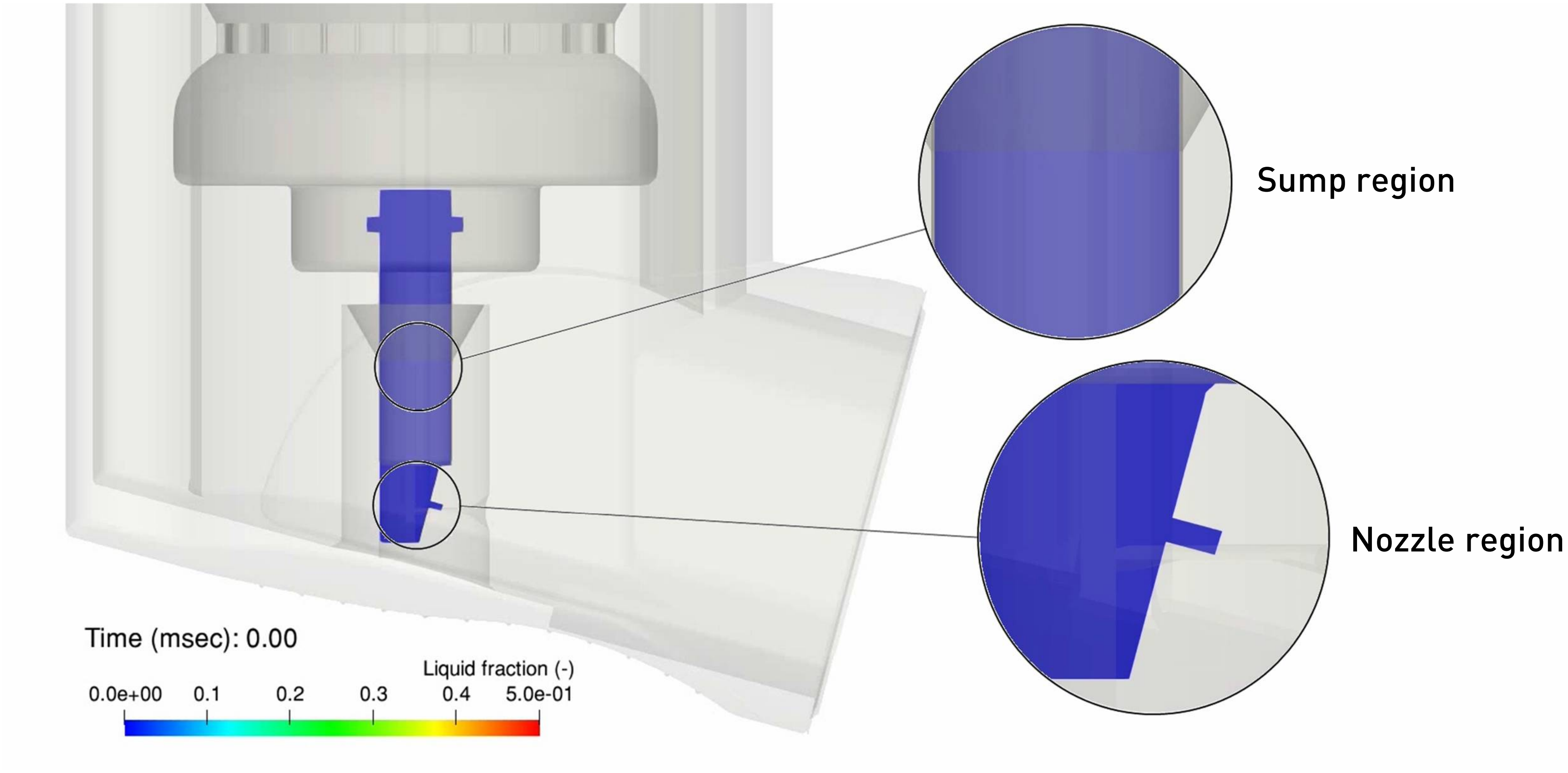
Challenges using OpenFOAM:

- **No compressible solver** with phase-change (was) available
- Standard phase-change models (strictly) valid only for **homogeneous conditions**
- **No heat transfer** and **no variable saturation pressure** available in the standard solver



X-ray visualization of internal flow structure in a pMDI.
(from Mason-Smith et al., 2017)

Modeling flashing in a pMDI (2/2)



Visualization of velocity field inside the pMDI computed using the CFD model (courtesy of Chiesi Farmaceutici S.p.A.)

Modeling deposition in human airways* (1/3)

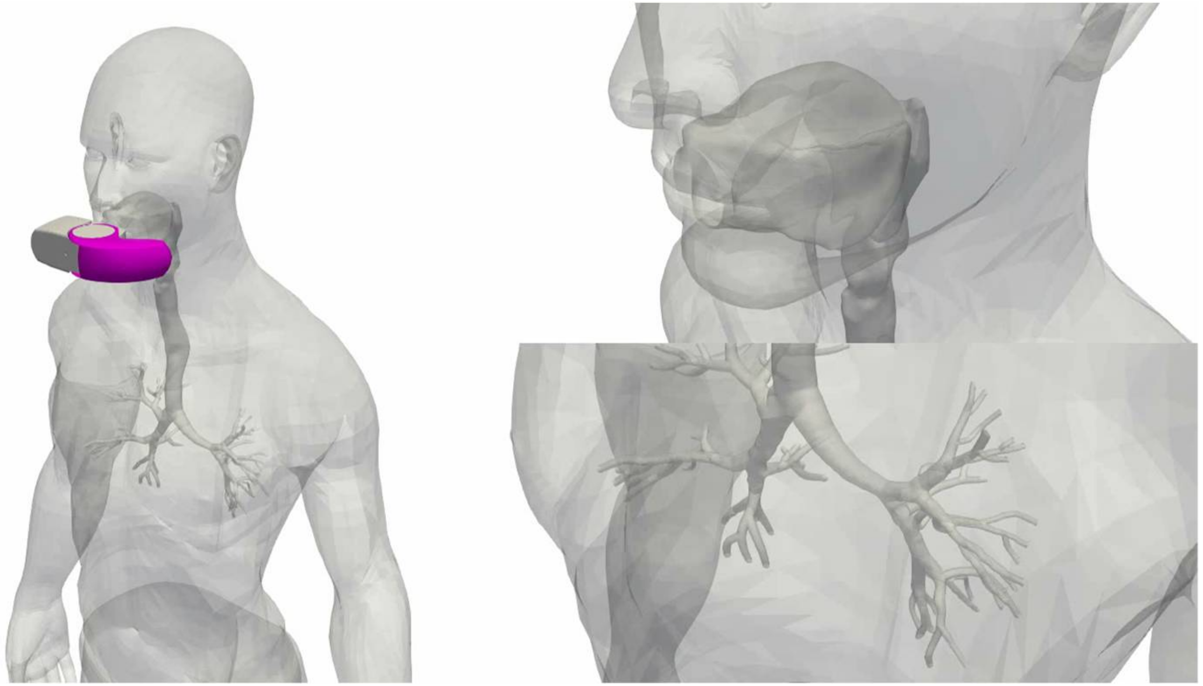
Despite the great amount of work in the simulation aerosol particle behavior inside the human respiratory system, certain **assumptions and approximations must be better clarified** and their quantitative impact on the simulated deposition process clearly analyzed.

Here these issues are addressed by means of **RANS simulations of the extra-thoracic airways evaluating** the impact of **steady flow** hypothesis, **turbulent dispersion** and **inflow conditions** on particles deposition.

Challenges using OpenFOAM:

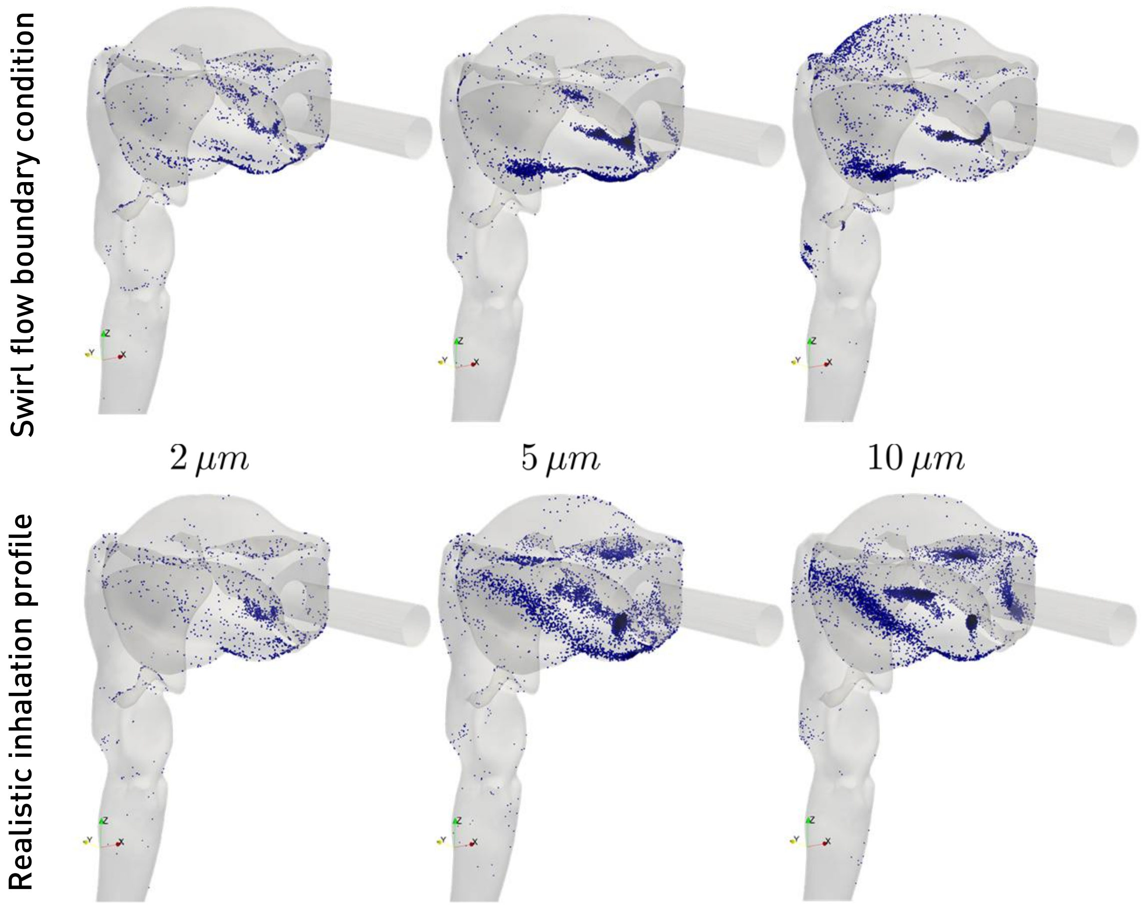
- **Mesh generation** with snappyHexMesh
- **Computational effort** and **robustness** of Lagrangian tracking
- Particles-**wall interaction**

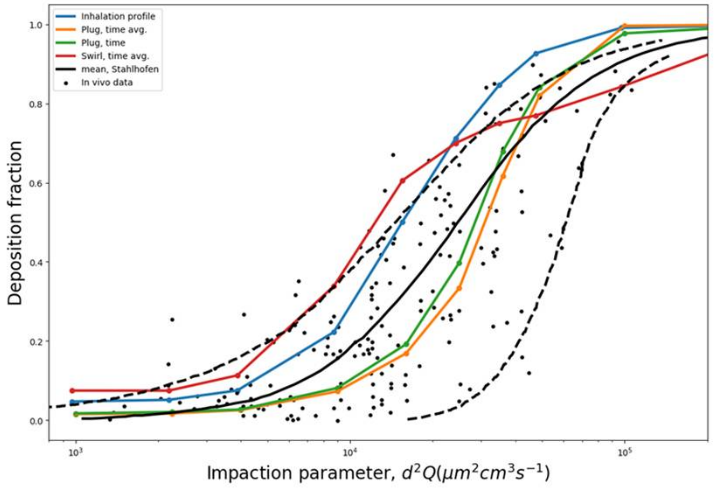
*in collaboration with SISSA-Trieste and G. H. Spasov (PhD candidate)

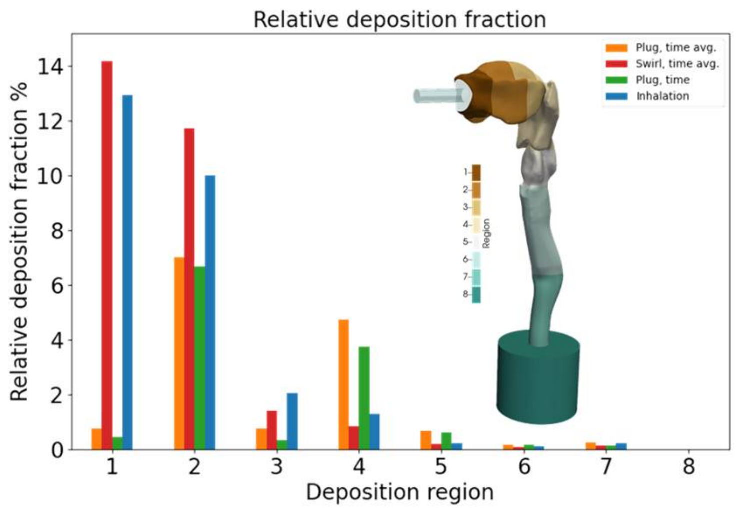


Animation of particles deposition in the human airways (courtesy of Chiesi Farmaceutici S.p.A.)

Modeling deposition in human airways (2/3)







Effect of boundary conditions and flow unsteadiness on deposition in the upper airways
(courtesy of Chiesi Farmaceutici S.p.A.)

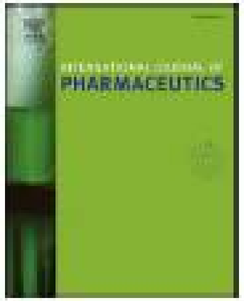
Modeling deposition in human airways (3/3)

International Journal of Pharmaceutics 629 (2022) 122331

Contents lists available at ScienceDirect

International Journal of Pharmaceutics

journal homepage: www.elsevier.com/locate/ijpharm



A critical analysis of the CFD-DEM simulation of pharmaceutical aerosols deposition in extra-thoracic airways

G.H. Spasov [a,c], R. Rossi [b], A. Vanossi [a,c], C. Cottini [d], A. Benassi [a,d,*]

[a] *International School for Advanced Studies (SISSA), Trieste, Italy*
[b] *RED Fluid Dynamics, Cagliari, Italy*
[c] *CNR-IOM, Consiglio Nazionale delle Ricerche - Istituto Officina dei Materiali, Trieste, Italy*
[d] *Chiesi Farmaceutici S.p.A., Parma, Italy*

RED Fluid Dynamics

...to surfboards design

Overview (1/2)

RED Team during experimental tests. Test locations (clockwise from top left): Portugal, Sardinia, France, Spain.

Overview (2/2)

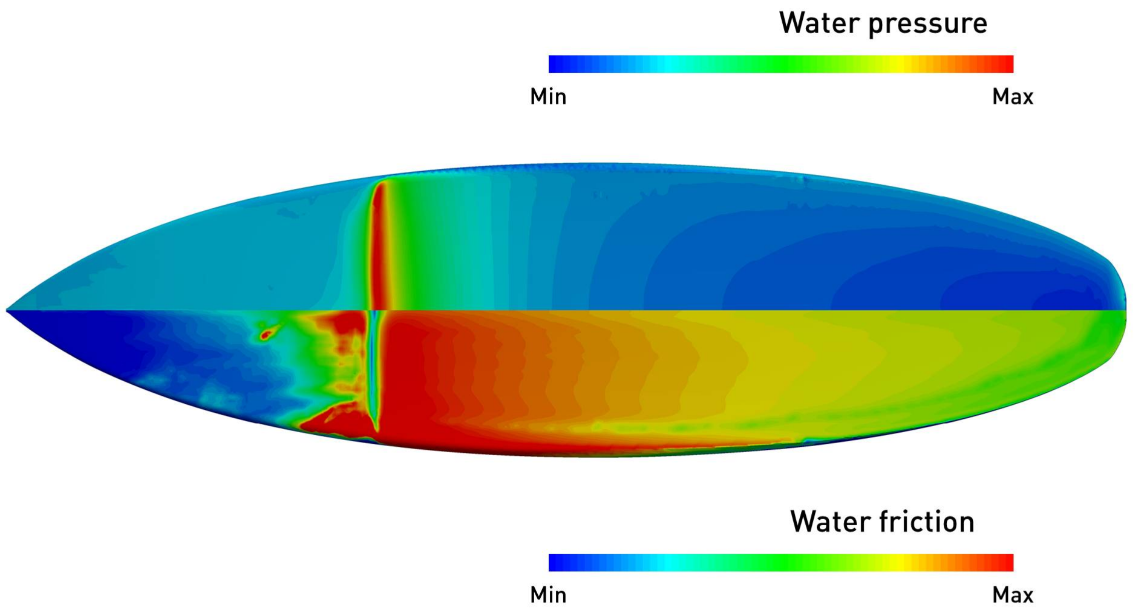
In 2017, we have introduced **for the very first time CFD simulations in the surfing industry** to allow **for predicting the basic performance of designs** before they are built.

Why OpenFOAM:

Despite being a **billion dollar industry**, the development of surfing gear is still **driven by experience** and **investments in technology are very limited**, especially in surfboards design

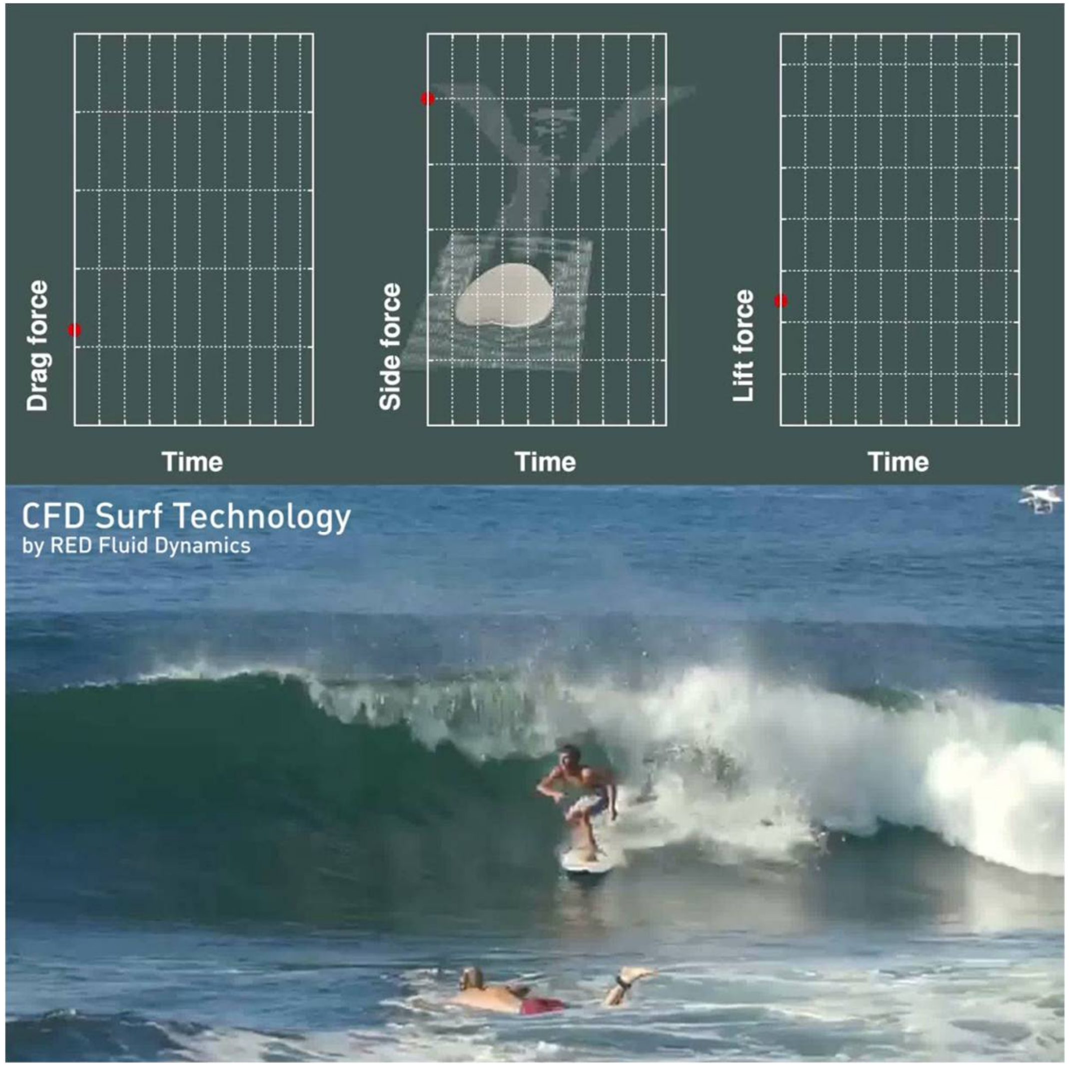
Challenges using OpenFOAM:

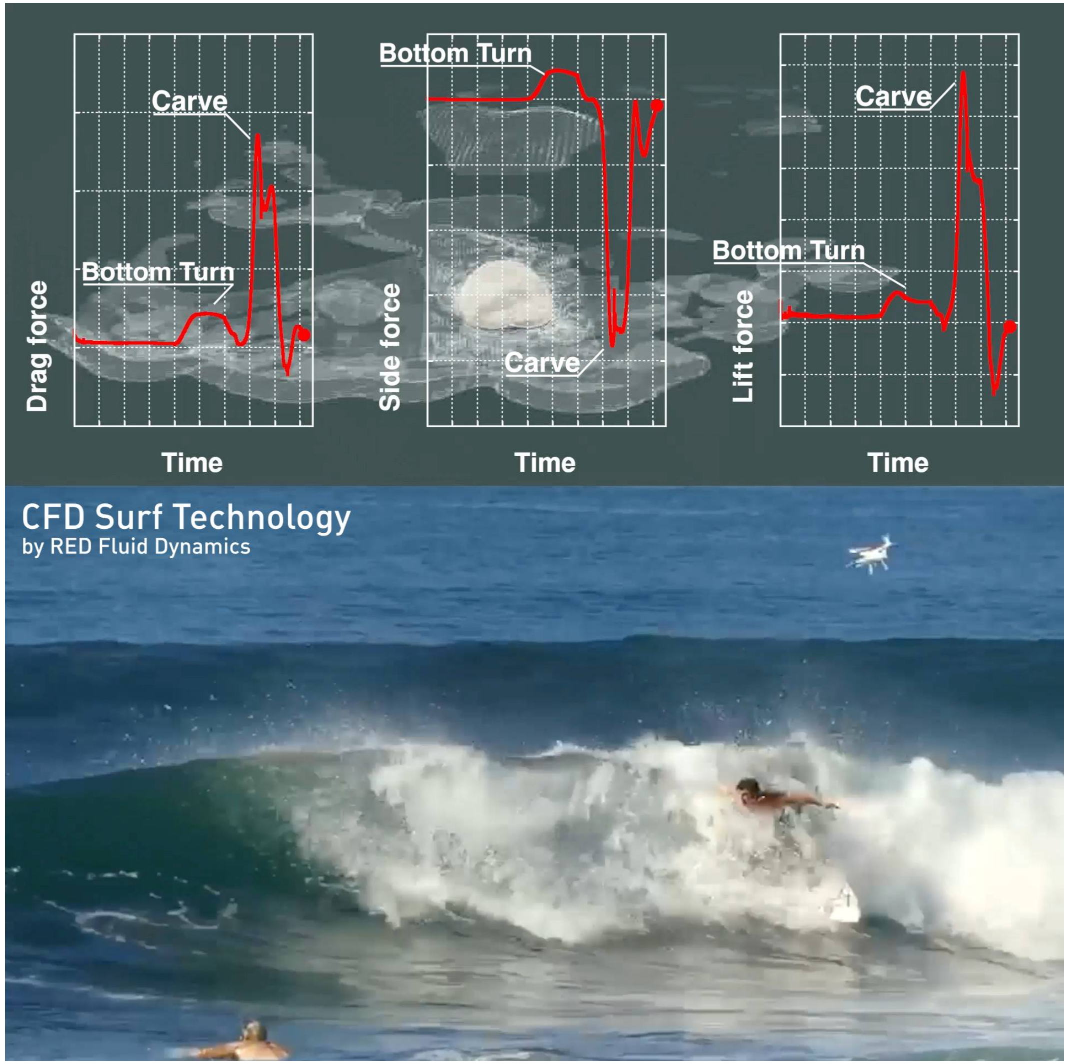
- Mesh generation and **layers coverage**
- Efficient **overset** technology
- Numerical **ventilation**



Static simulation of a surfboard during cruise (courtesy of Sequoia Surfboards)

Throwing some (Open)FOAM





Dynamic simulation of a surfboard during a sequence of maneuvers (courtesy of Sequoia Surfboards)

RED Fluid Dynamics

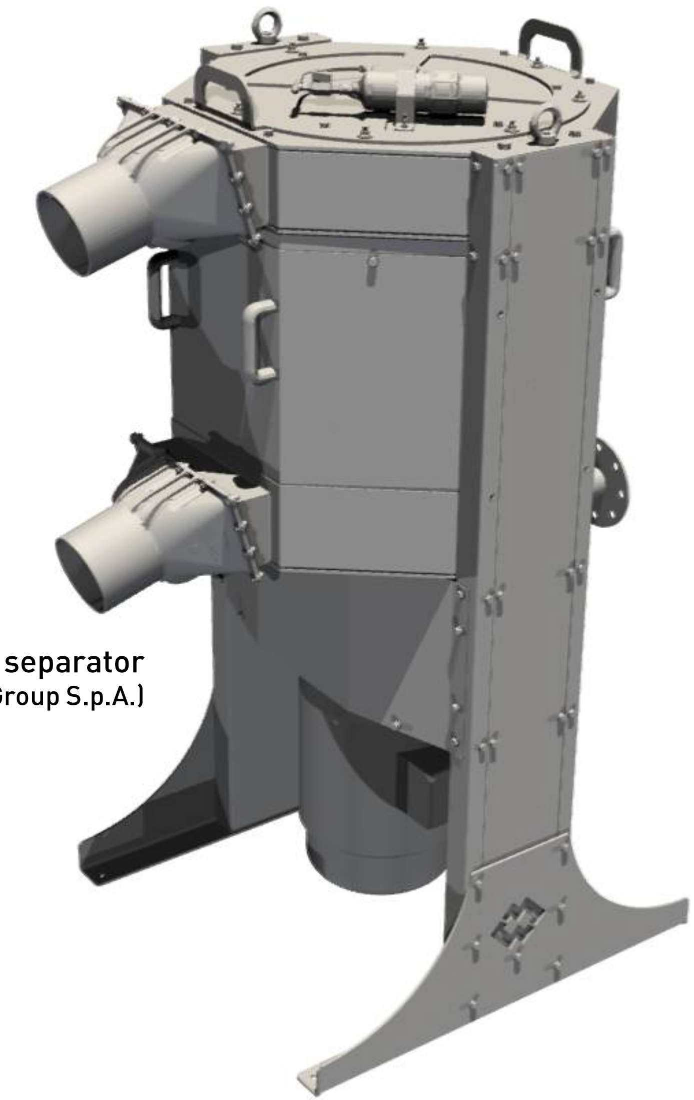
Exploiting the power of OpenFOAM: a case study

Exploiting OpenFOAM: a case study (1/6)

In this case study, OpenFOAM was used for the simulation of the multiphase flow inside a **centrifugal liquid/solid separator**.

The simulation of the separator required the modeling of **many complex flow features**:

- Complex geometry
- Turbulent flow regime
- Multiphase flow with gas, liquid and solid phase
- Rotating device
- **Microfilter** for solid/liquid separation



SEPCOM MFT separator
(courtesy of WAM Group S.p.A.)

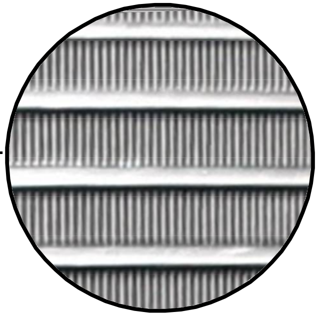
Exploiting OpenFOAM: a case study (2/6)

Confidential

Animation of the multiphase flow inside the SEPCOM MFT separator
(courtesy of WAM Group S.p.A.)

RED Fluid Dynamics

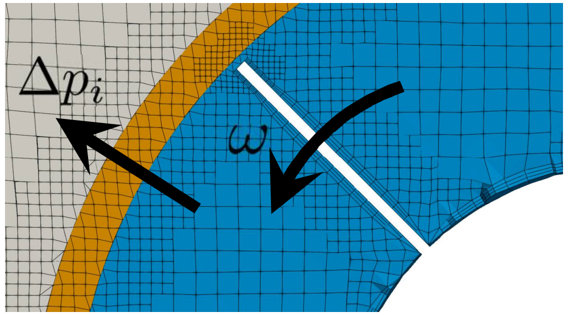
Exploiting OpenFOAM: a case study (3/6)



Microfilter modeling

- Porous media (filter/liquid interaction)
- **Virtual filter** (filter/solid interaction)

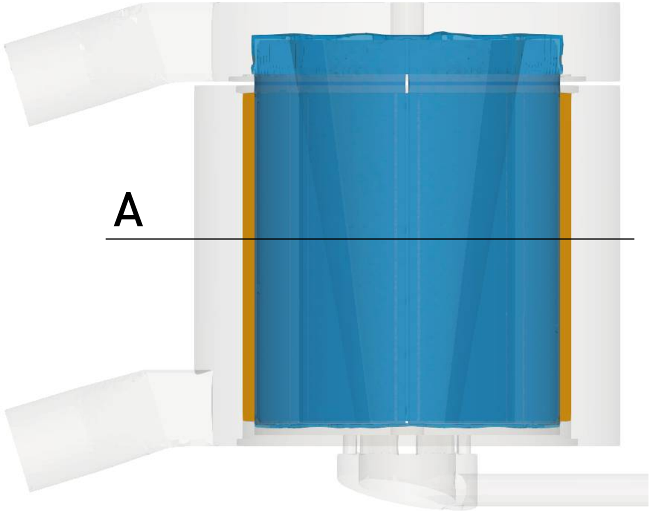
Section A



Porous media

MRF static region

MRF rotating region



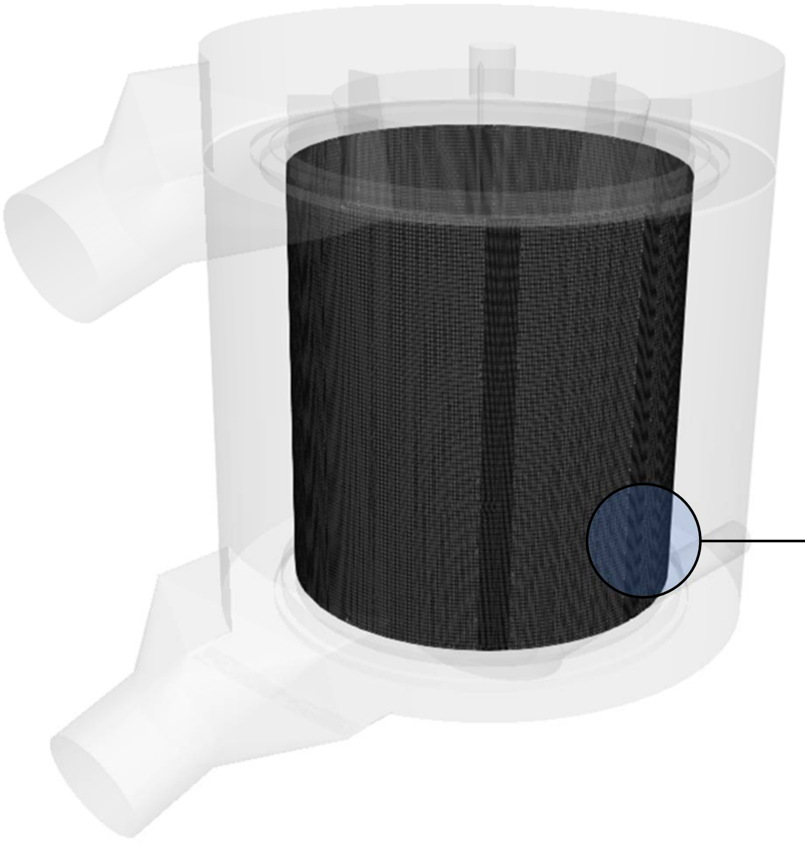
Exploiting OpenFOAM: a case study (4/6)

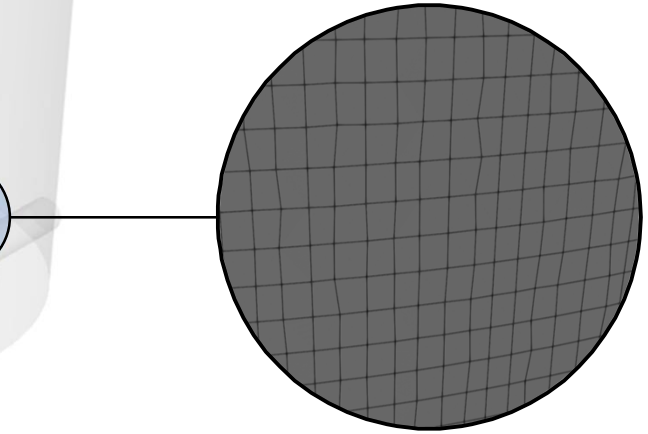
The virtual filter was implemented by adding a **new `patchInteractionModel`** within the **`Kinematic` submodels** of the **`intermediate` Lagrangian** library.

```
// Calculate impact angle between particle and filter patch (0=tangential, 90=perpendicular)
p.filterHitAngle() = asin((U & nw)/(mag(U)*mag(nw)))*180.0/Foam::constant::mathematical::pi;

// Let particle move through the filter if diameter smaller or equal than filter size,
// number of hits larger or equal than threshold and hit angle larger or equal than threshold
if (p.d()<=patchData_[patchi].fsize() && p.filterHits()>=patchData_[patchi].fminHits() &&
    p.filterHitAngle()>=patchData_[patchi].fminAngle())
{
    p.filterHits() += 1.0;

    return false;
}
```



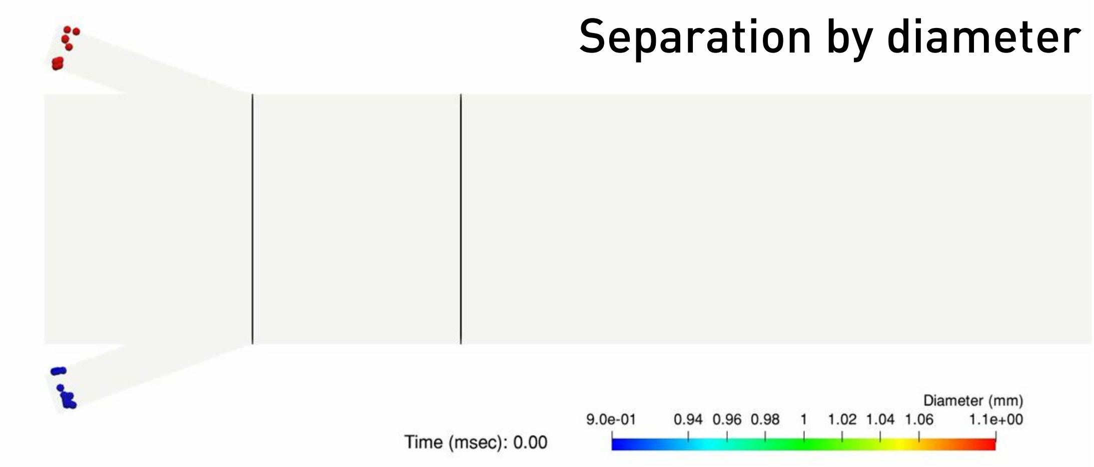


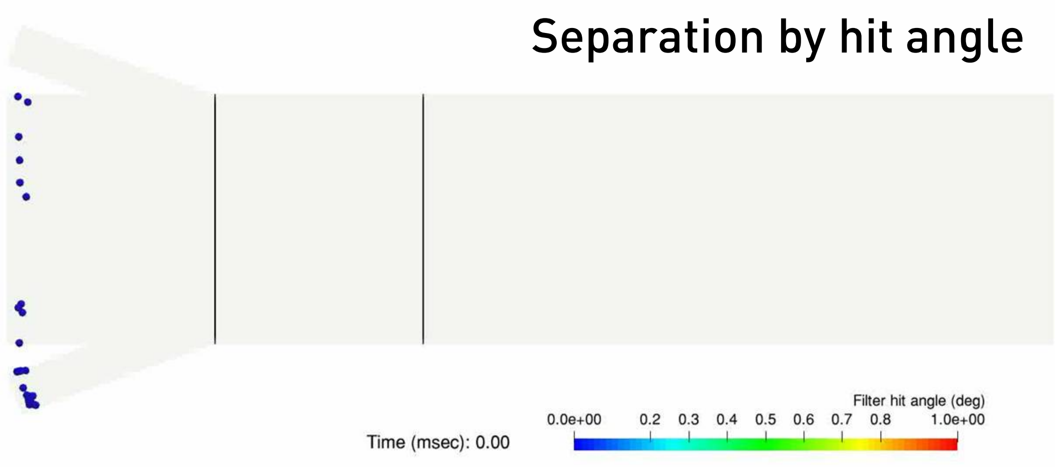
Virtual filter patch inside the separator.

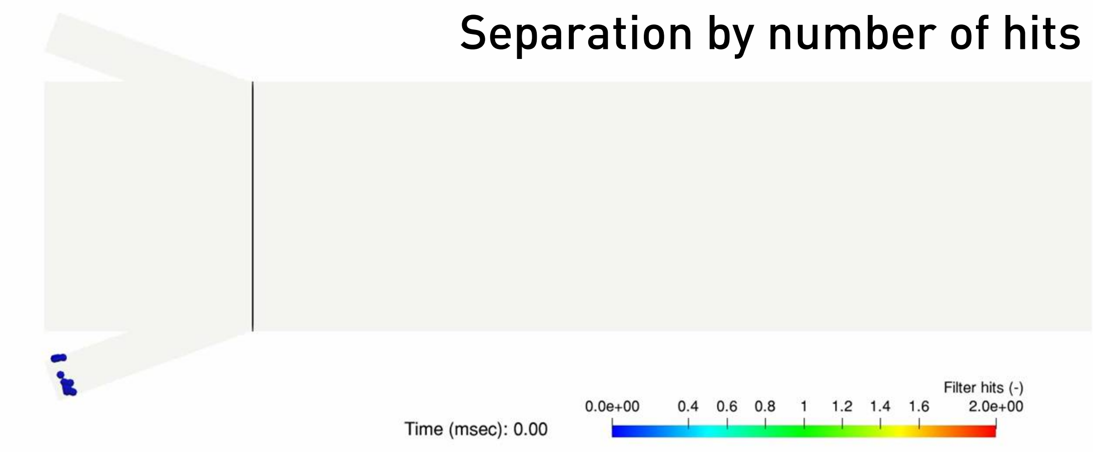
Virtual filter parameters

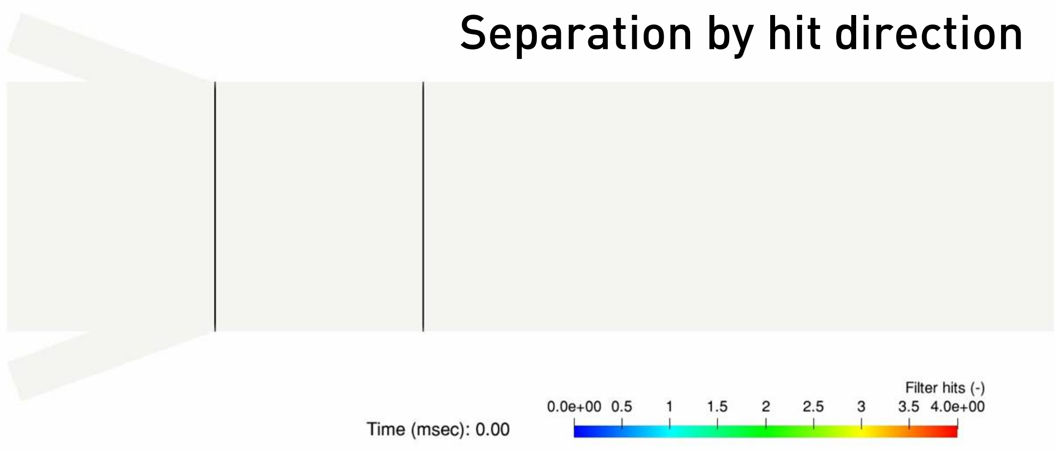
- Average slot opening
- Minimum hit angle
- Minimum number of hits
- Hit direction

Exploiting OpenFOAM: a case study (5/6)









Testing the virtual filter model in a simplified flow and particles setup.

RED Fluid Dynamics

Exploiting OpenFOAM: a case study (6/6)

Confidential

Visualization of the liquid and particles interaction with the rotor and the microfilter
Inside the separator (courtesy of Wam Group S.p.A.)

RED Fluid Dynamics

Final remarks

Final remarks (1/3)

Pros and cons:

- **Multidisciplinary consulting** only possible thanks to **strong academic background**
- Very often **new features discovered by chance**
- **Robustness** should be improved
- We get paid to learn
- **Never gets boring**

Wishlist:

- **Solvers unification by major flow physics** (as in upcoming 11.0 release)
- Full layer coverage with snappyHexMesh (i.e. Engys like)
- Improved Lagrangian tracking algorithm
- Efficient overset technology

Final remarks (2/3)

This is an early access version, the complete PDF, HTML, and XML versions will be available soon.

Open Access Review

A Review of Laboratory and Numerical Techniques to Simulate Turbulent Flows

by Simone Ferrari [1,*], Riccardo Rossi [2] and Annalisa Di Bernardino [3]

[1] Department of Civil-Environmental Engineering and Architecture (DICAAR), University of Cagliari, 09123 Cagliari, Italy
[2] RED Fluid Dynamics, 09127 Cagliari, Italy
[3] Physics Department, Sapienza University, 00185 Rome, Italy
[*] Author to whom correspondence should be addressed.

Academic Editor: Artur Blaszczuk

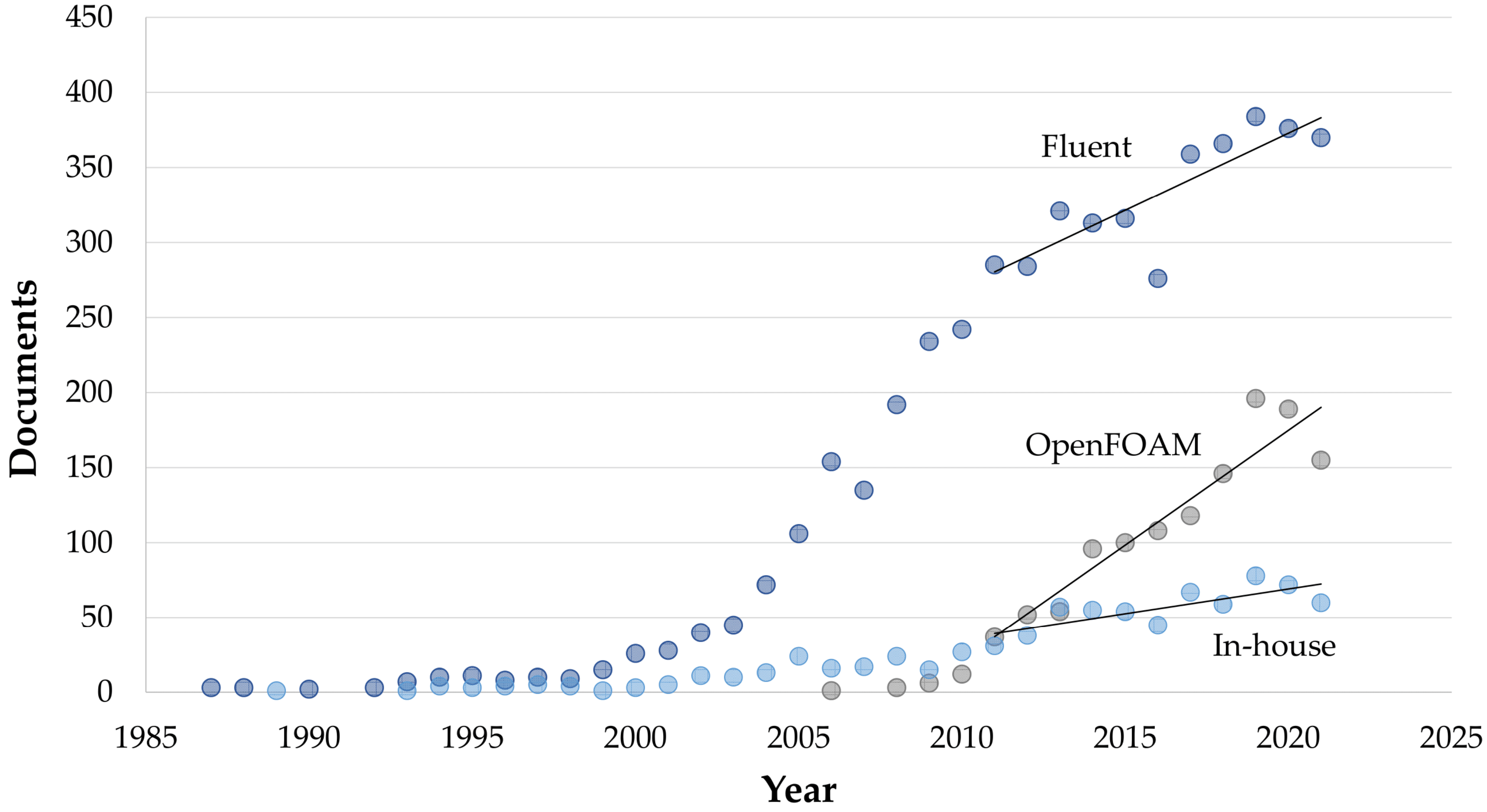
Energies **2022**, *15*(20), 7580; https://doi.org/10.3390/en15207580 (registering DOI)

Received: 18 July 2022 / Revised: 23 September 2022 / Accepted: 10 October 2022 / Published: 14 October 2022

(This article belongs to the Section **A3: Wind, Wave and Tidal Energy**)

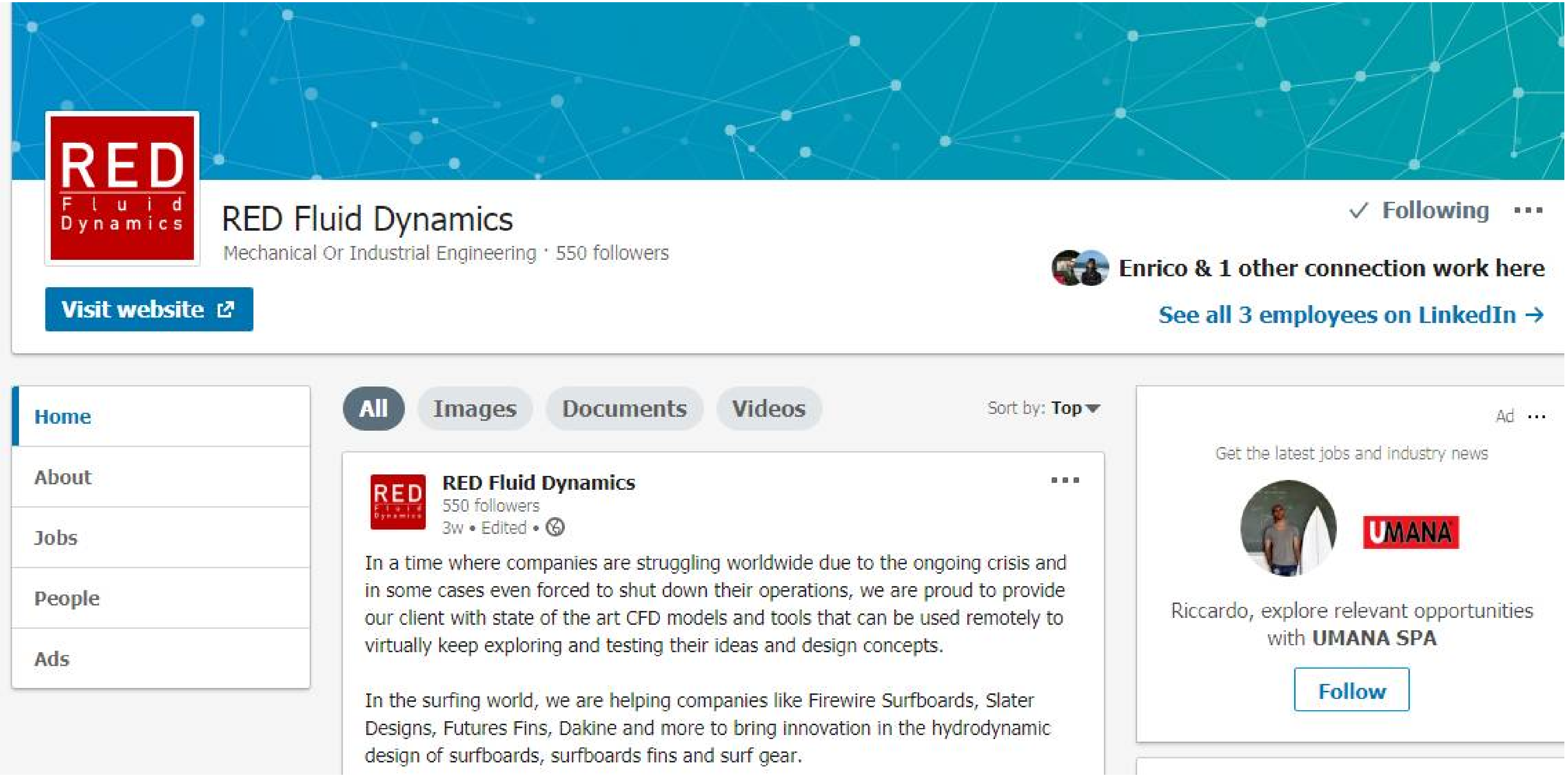
Download PDF Citation Export

Final remarks (3/3)



Statistics of papers published in indexed scientific journals with turbulence, simulation, and software types in title, abstract, or keywords (source: Scopus.com).

Follow us!



RED Fluid Dynamics