Contributing to OpenFOAM Foundation Release – an effective code development strategy

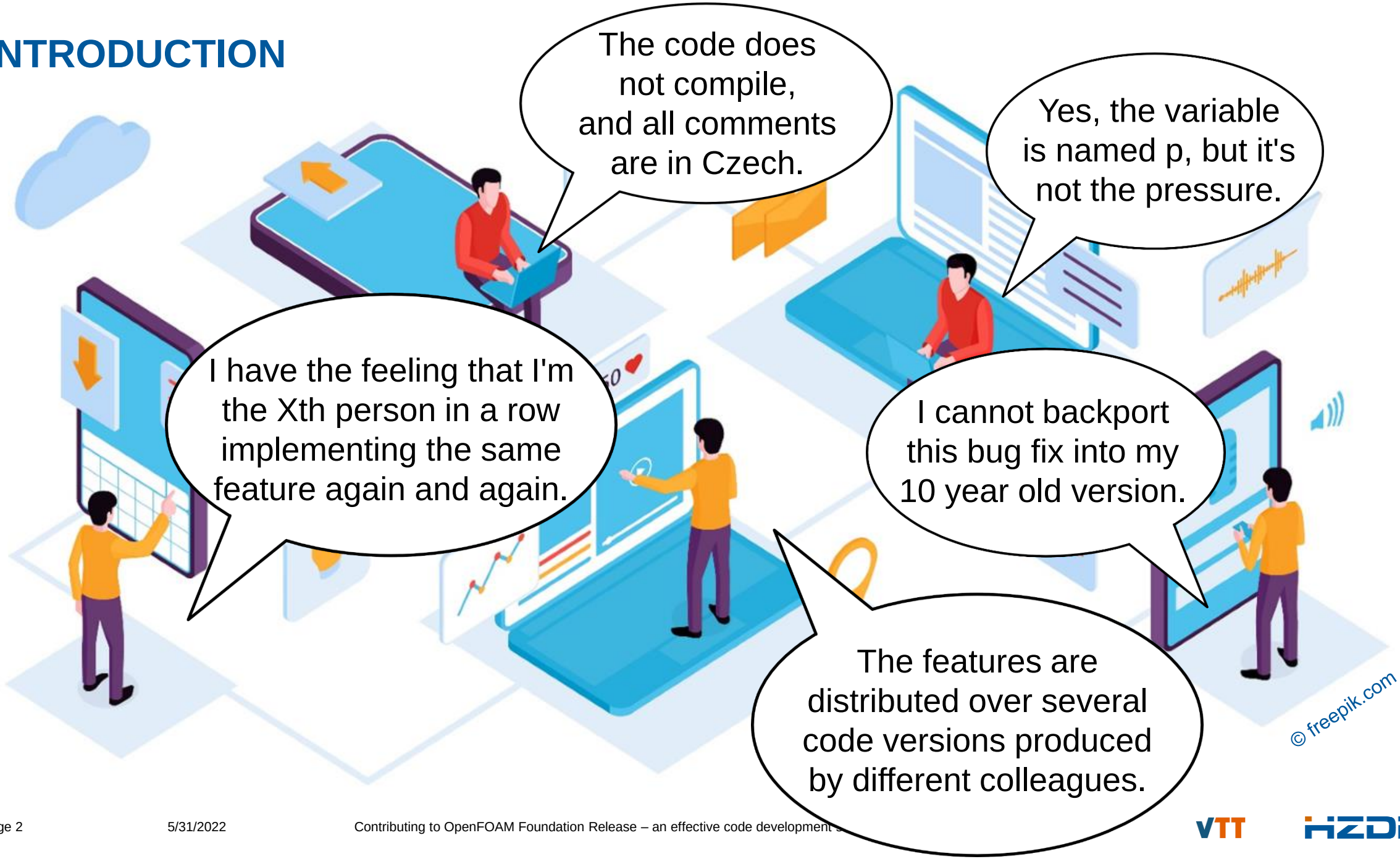
F. Schlegel[1] and J. Peltola[2]

[1)] Helmholtz-Zentrum Dresden-Rossendorf, Germany

[2)] VTT Technical Research Center, Finland

Institut für Fluiddynamik · Computational Fluid Dynamics · Dr. Fabian Schlegel · f.schlegel@hzdr.de · https://hzdr.de/openfoam

INTRODUCTION



INTRODUCTION

(Cost, Time) efficiency

Sustainability

Costs for CFD Software

Start writing from scratch

- File I/O
- Converters (file formats)
- Meshing
- Discretisation schemes
- Matrix solvers
- Parallelization
- Equations and models
- Documentation
- …

Costs for CFD Software

When start sharing it with others

Release management

- deployment, packaging, installation, …

Quality management

- bugs, re-factoring, dependencies, validation/verification, …

Software management

- API, new features, documentation, …

Communication management

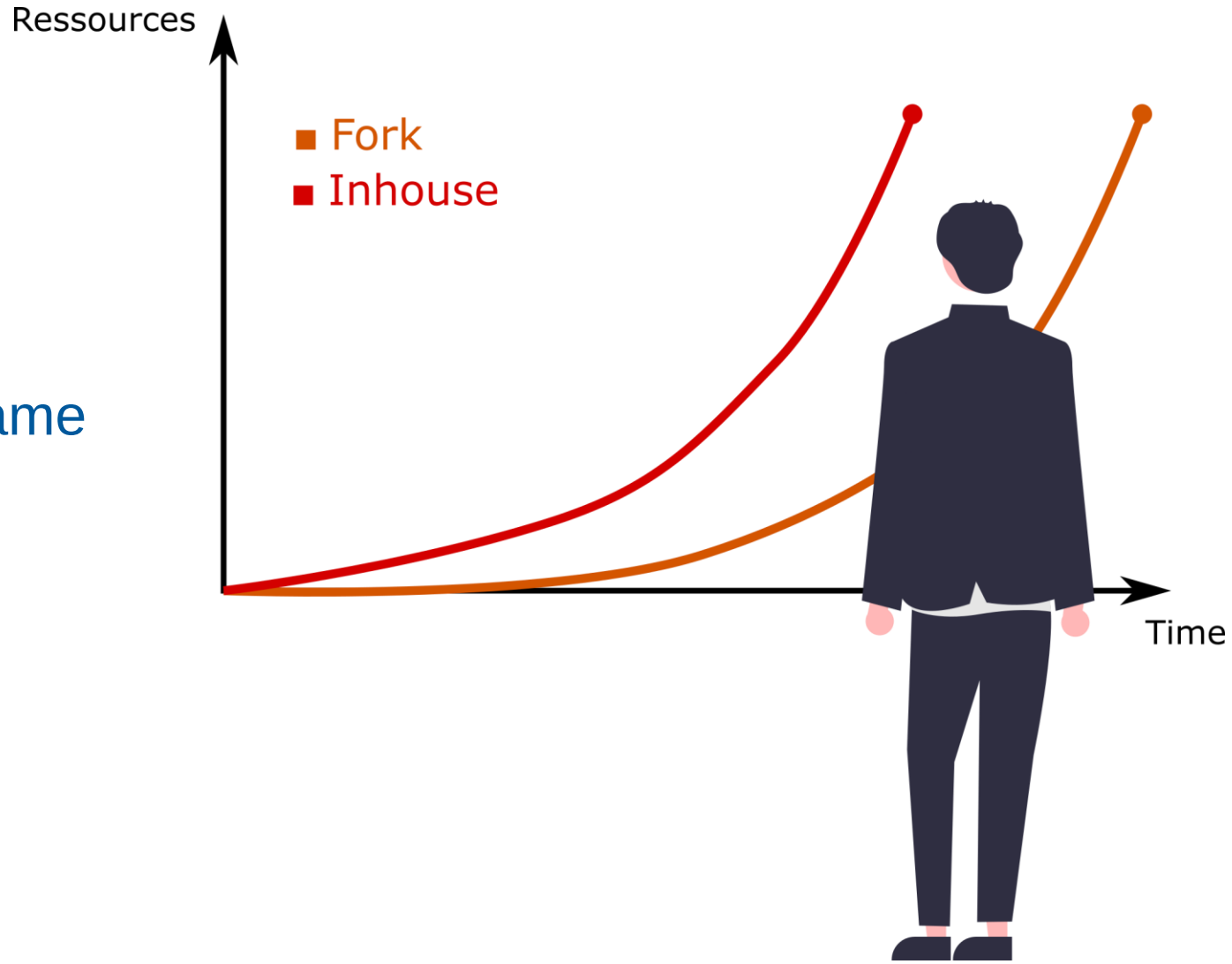
- coordination, advertisement, conflict management, …

Knowledge management

Costs for CFD Software

Providing OSS Software

- **Cost** will **rise** over time,
  and you have to cover them
- **Forking** gives **head start**
- **Long-term costs** will be the same
- Limits industry transfer

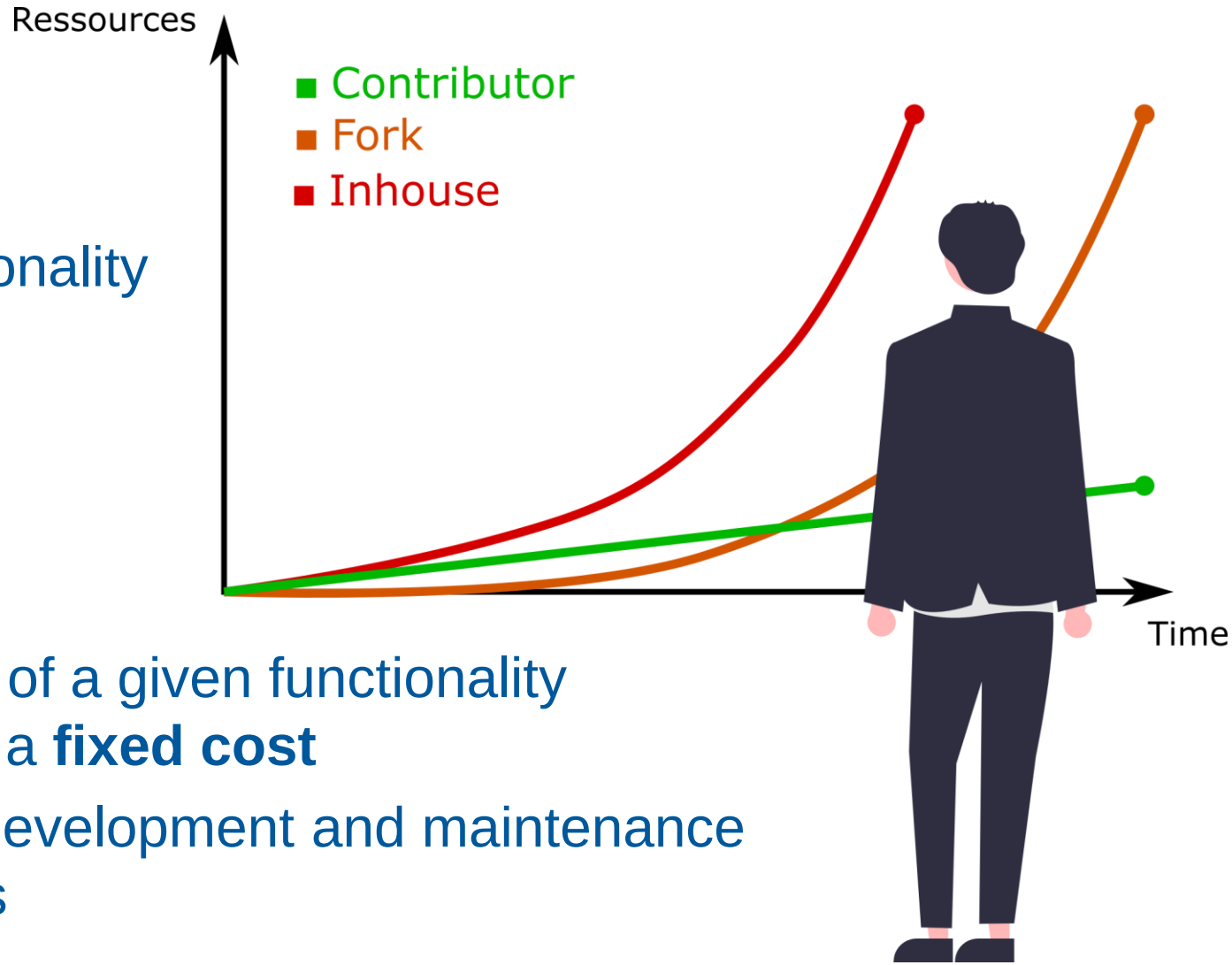


Costs for CFD Software

User of OSS Software

No license fees but still costs for

- **Development** of new functionality
- **Maintenance** and redesign
- Hardware
- **User know how**

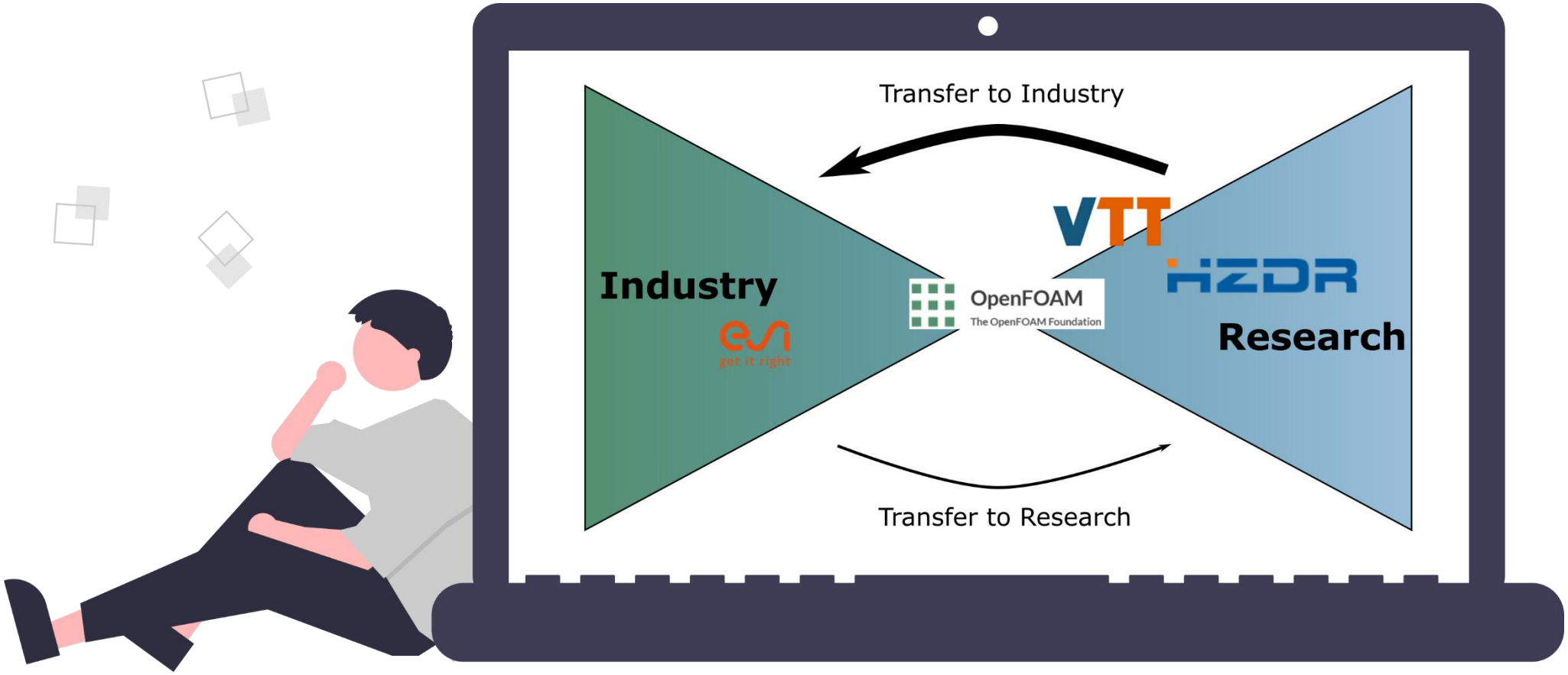


Development and maintenance of a given functionality and implementation is essentially a **fixed cost**

► key to cost efficiency: spread development and maintenance cost across multiple organizations

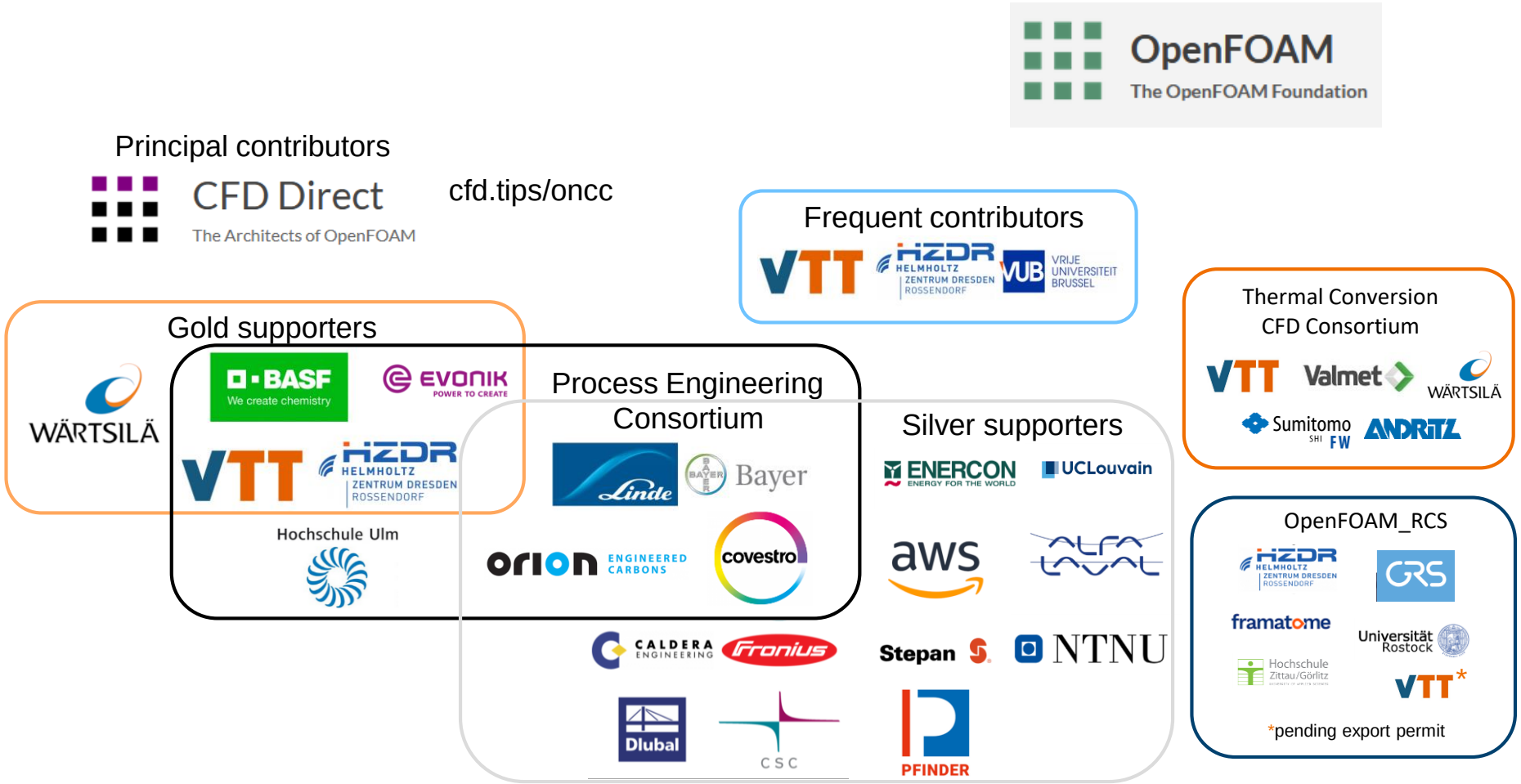
Contribution to OpenFOAM

Why OpenFOAM Foundation release



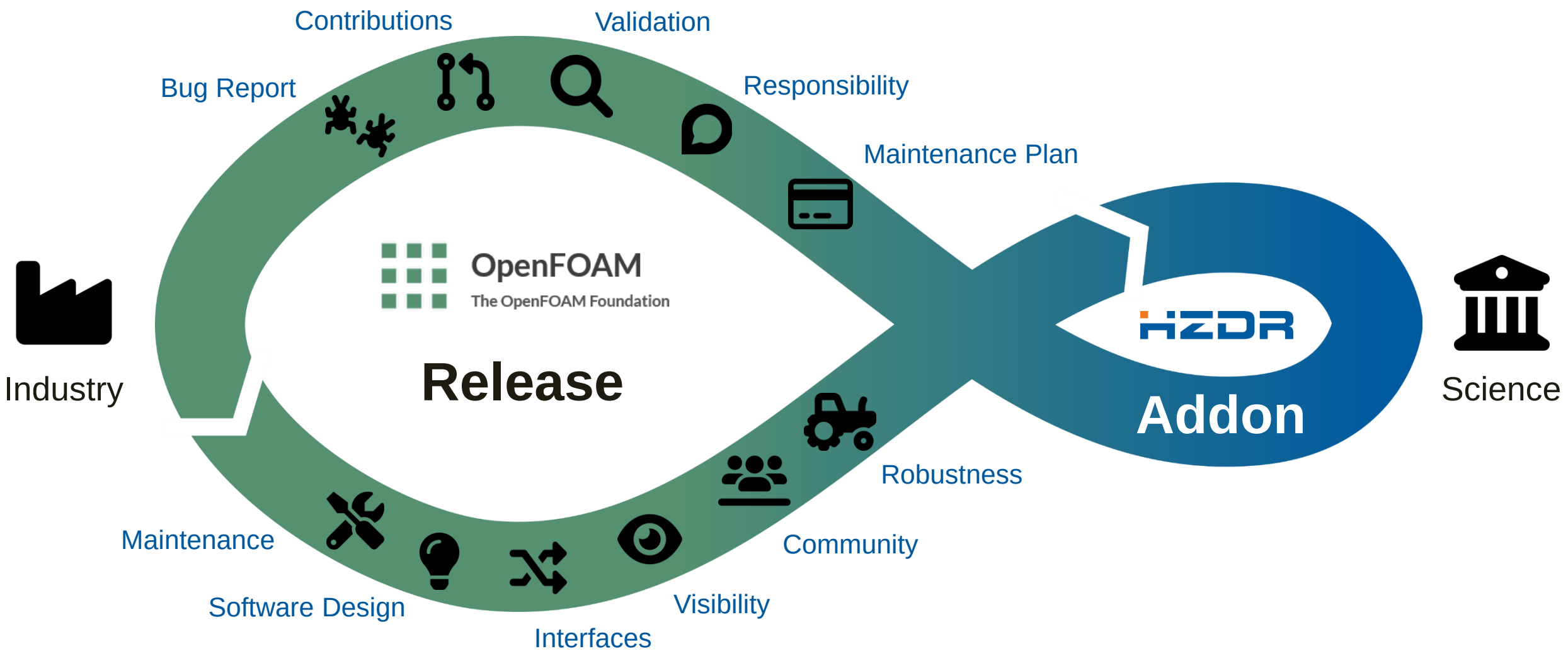
CONTRIBUTION TO OPENFOAM

ACTIVE COLLABORATION AROUND OPENFOAM FOUNDATION



CONTRIBUTION TO OPENFOAM

CONTRIBUTOR WITH ADDON



Contribution to OpenFOAM

Software Quality

Feasibility Prototype
- Validate individual features
- Check feasibility
- Extremely important

Preliminary Product Prototype
- Interplay of components
- Identify limitations and shortcomings
- Selected features

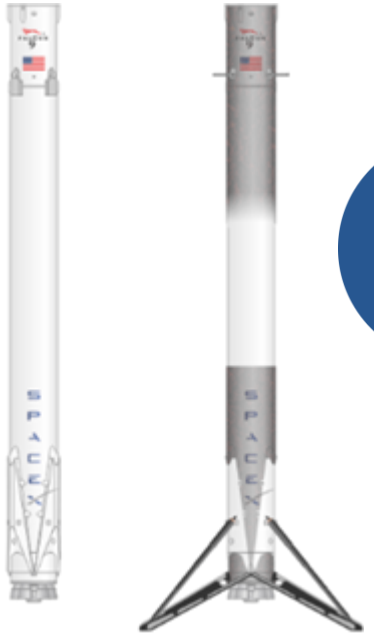
Demonstration Prototype
- Most of the features and functions
- Start user testing

Production Software
- All features
- Fully optimized
- Production ready

Core Maintainers

Grasshopper test vehicle

Falcon 9R Test Vehicle



Falcon 9 FT

Contribution Example

2015

First line of code for class method

2016

Feasibility prototype

“Working iClassMethod implementation for HZDRmultiphaseEulerFoam and HZDRreactingMultiphaseEulerFoam including 0D and bubbleColumn test cases.”

Preliminary Product Prototype

Redesign after C++ Hackathon

2017

Collaboration with VTT Finland

Mass transfer, boiling, encapsulation

Contribution Example

2017

Demonstration Prototype
commit 3e577d8515dccaa079827db9d40a220b36a55647

2018

Bugs, new kernels, speed improvements by CFDDirect and VTT

New directory for model tests and validation

2019

Fractal shape modelling
commit cfbb389fd3eafa0c901ae151741868379b9c1c2b

Contribution Example

2020

Side project: porting class method to GPU

ThermophysicalTransportModels: New library to handle the transport of energy and species

thermophysicalModels: Added new tabulated equation of state, thermo and transport models

2021

Technical guide on "Disperse Multiphase Flows in OpenFOAM"

Production software
Process Engineering Consortium, Alfa Laval, Research Institutes worldwide

multiphaseEulerFoam: Added moving-mesh
support for both cell- and face-momentum algorithms

Contribution Example

2022

Long term commitment of HZDR to support maintenance and new feature development of population balance, e.g., new functionObject "moments“
commit 36c565b9bf5d0f318e0e908c2097fd18f14d1e10

Added solid particle coalescence and breakup model (Adachi et al. ,1994, Kusters, 1991), spherical particle lift force model by Saffman-Mei (1992), and tutorials
commit 36c565b9bf5d0f318e0e908c2097fd18f14d1e10

Non-Conformal Coupled (NCC): Conservative coupling of non-conforming patches

Alternative Way – OpenFOAM_RCS

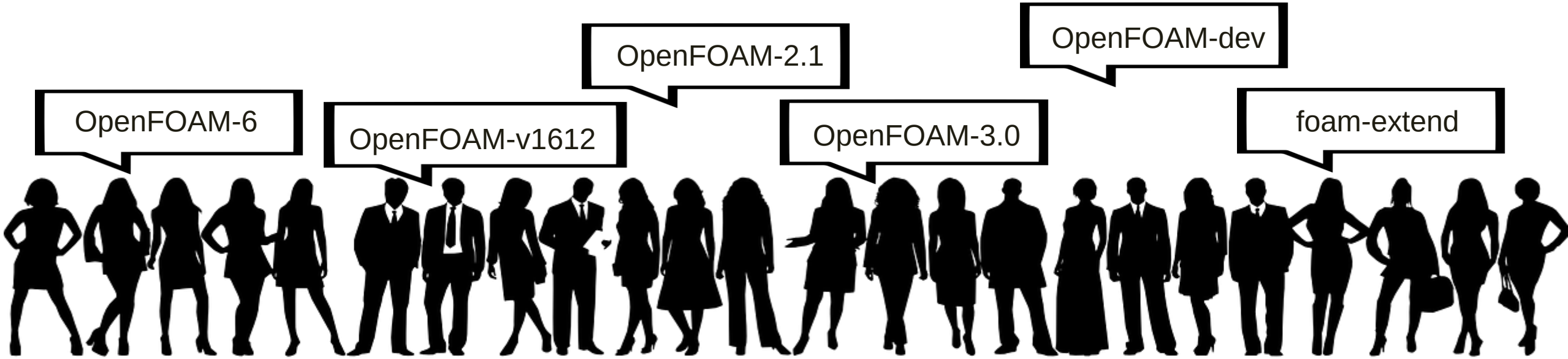
Gefördert durch:

Bundesministerium
für Umwelt, Naturschutz, nukleare Sicherheit
und Verbraucherschutz

aufgrund eines Beschlusses
des Deutschen Bundestages

OpenFOAM_RCS Project

OpenFOAM Versions in German CFD Alliance



Coordinator ▶ Helmholtz-Zentrum Dresden-Rossendorf

Code ▶ OpenFOAM Foundation Release

https://hzdr.de/openfoam-rcs

OpenFOAM_RCS Project

Main Goals

- Provide a **sustainable environment** for code and setups, that cannot go into the OpenFOAM development line
  - export control limitations
  - software still a prototype
  - different design philosophy
- Foster **collaborative code** and **case development** based on a reference OpenFOAM installation
- Keep simulation setups and code alive for future projects
- Build up a **validation data base** for nuclear safety applications

OpenFOAM_RCS Project

Repositories

Software

- GPL license
- Source Code
  - ► reference solution functionObject
- Test setups
  - ► for physical property evaluation
- Tutorials
  - ► HZDR Baseline model

Restricted

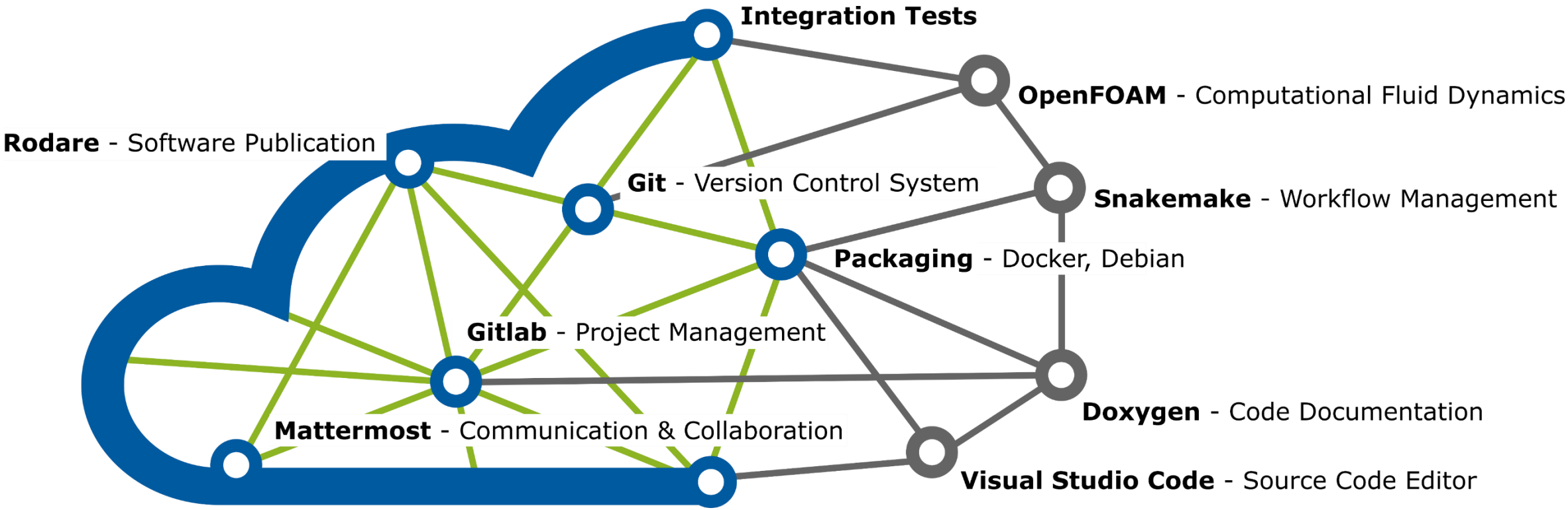
- Proprietary license
- Simulation setups
  - ► periodic hill channel
- Validation data base

Gitlab CI/CD

„pipelines“ for functional tests & validation

OpenFOAM_RCS Project

HELMHOLTZ CLOUD



VTT HZDR

OpenFOAM_RCS Project

Partners

Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH

Ruhr University Bochum, Plant Simulation and Safety

University Rostock, Lehrstuhl für Strömungsmechanik

Pending Partners

- **Technical University of Munich** – Research Neutron Source Heinz Maier-Leibnitz (FRM II)
- **Hochschule Zittau-Görlitz**
- **University of Stuttgart** – Institute of Nuclear Technology and Energy Systems
- **Technical University of Munich** – Chair of Energy Systems
- **VTT Technical Research Centre of Finland Ltd.**

OpenFOAM_RCS Project

Running Projects

Coupling of external **thermophysical properties** library with OpenFOAM

► GPL License requires coupling via sockets

Highly-automated **workflow management** tool for validation database

► Snakemake, Python-based, workflow tool from bio-informatics

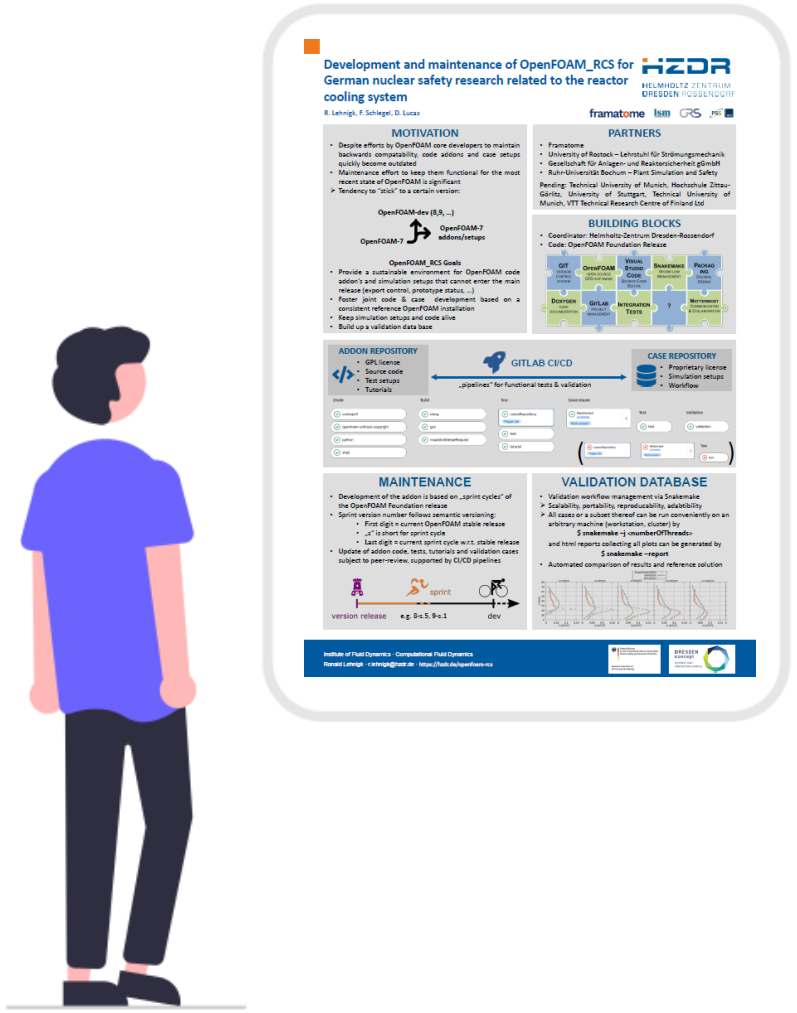
OpenFOAM_RCS Project

Poster

We are looking forward to further discussions in front of our poster

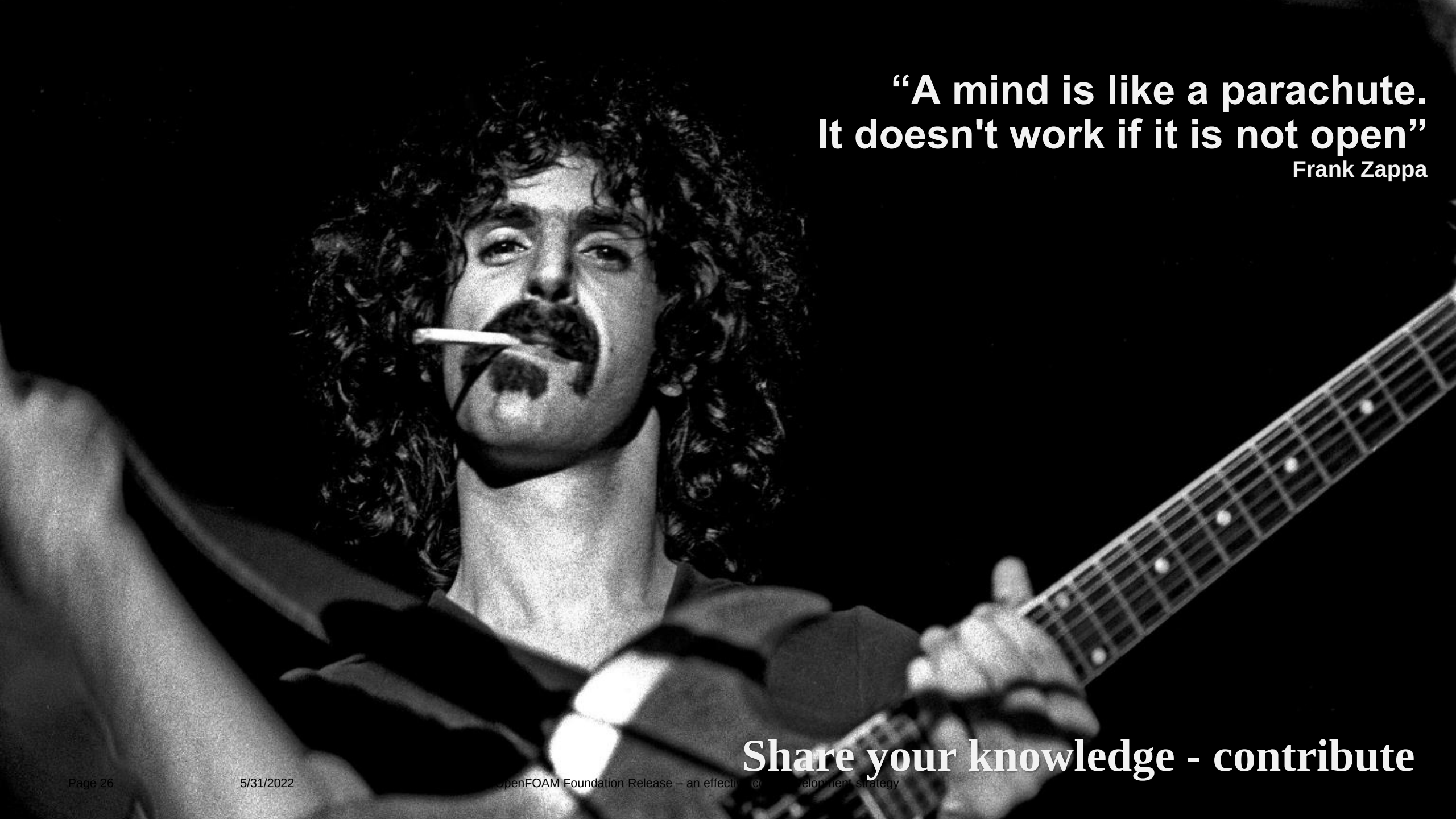
“Development and maintenance of OpenFOAM_RCS for German nuclear safety research related to the reactor cooling system”

by Lehnigk, R.; Lucas, D. and Schlegel, F.



SUMMARY

- Do not maintain a separate fork, **contribute your developments**
- Contributions to OpenFOAM Foundation are **possible and valuable**
- Contributions have short term costs, but **long term gains**
- Contributions are the most sustainable way


“A mind is like a parachute.
It doesn't work if it is not open”
Frank Zappa
Share your knowledge - contribute