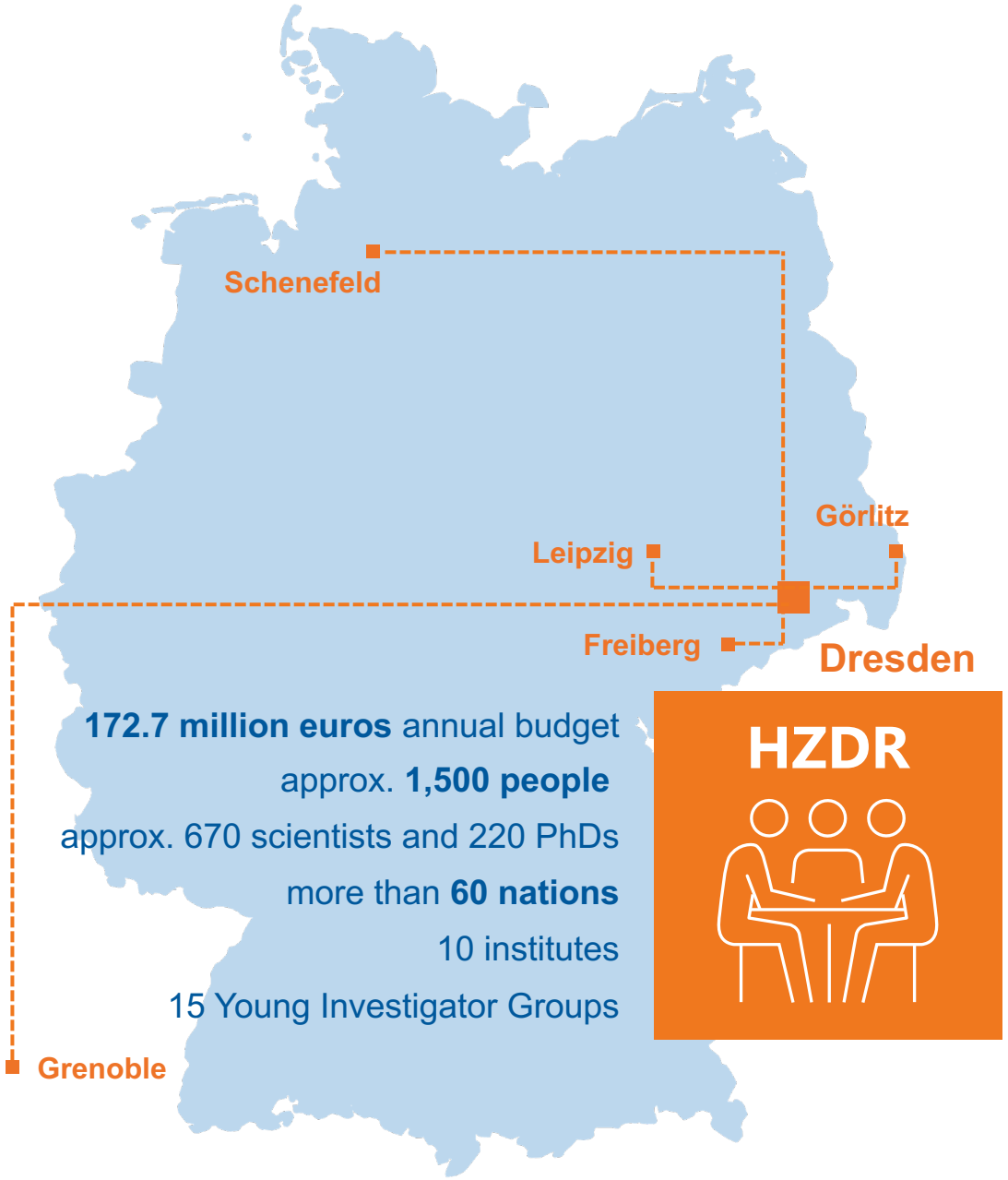
HELMHOLTZ ZENTRUM
DRESDEN ROSSENDORF

# Sustainable Code Development and Cases Management with OpenFOAM Foundation Release

## Free OpenFoam VKI Seminar

Institute of Fluid Dynamics – Department of Computational Fluid Dynamics – Dr. Fabian Schlegel – www.hzdr.de/multimorph


HZDR
HELMHOLTZ ZENTRUM
DRESDEN ROSSENDORF
SCANNING
G17
Q11
L21
R12
X1
X2
X3
X4




**172.7 million euros** annual budget

approx. **1,500 people**

approx. 670 scientists and 220 PhDs

more than **60 nations**

10 institutes

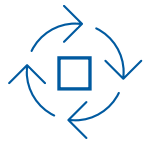
15 Young Investigator Groups

**HZDR**

# WHO ARE WE?

# INSTITUTES AND CENTRAL DEPARTMENTS

| | | | | |
|---|---|---|---|---|
| Institute of Fluid Dynamics | Institute of Ion Beam Physics and Materials Research | Institute of Theoretical Physics | Institute of Radiopharmaceutical Cancer Research | Department of Information Services and Computing |
| Institute of Resource Ecology | Institute of Radiation Physics | Dresden High Magnetic Field Laboratory | Institute of Radiooncology – OncoRay | Department of Research Technology |
| Helmholtz Institute Freiberg for Resource Technology | | CASUS – Center for Advanced Systems Understanding | | |



Energy

Matter

Health



DRESDEN concept HZDR

# Institute of Fluid Dynamics

**Departments** in Institute of Fluid Dynamics

**Computational Fluid Dynamics**

*Dirk Lucas*

**Experimental Thermal Fluid Dynamics**

*Uwe Hampel*

**Magneto-hydrodynamics**

*Sven Eckert*

**Transport Processes at Interfaces**

*Kerstin Eckert*

**Groups** in Department of Computational Fluid Dynamics

**OpenFOAM modelling of multiphase flows**

*Fabian Schlegel*

**Bubbles go with the turbulent flows**

*Tian Ma*

**Flotation and reactive multiphase flows**

*Roland Rzehak*

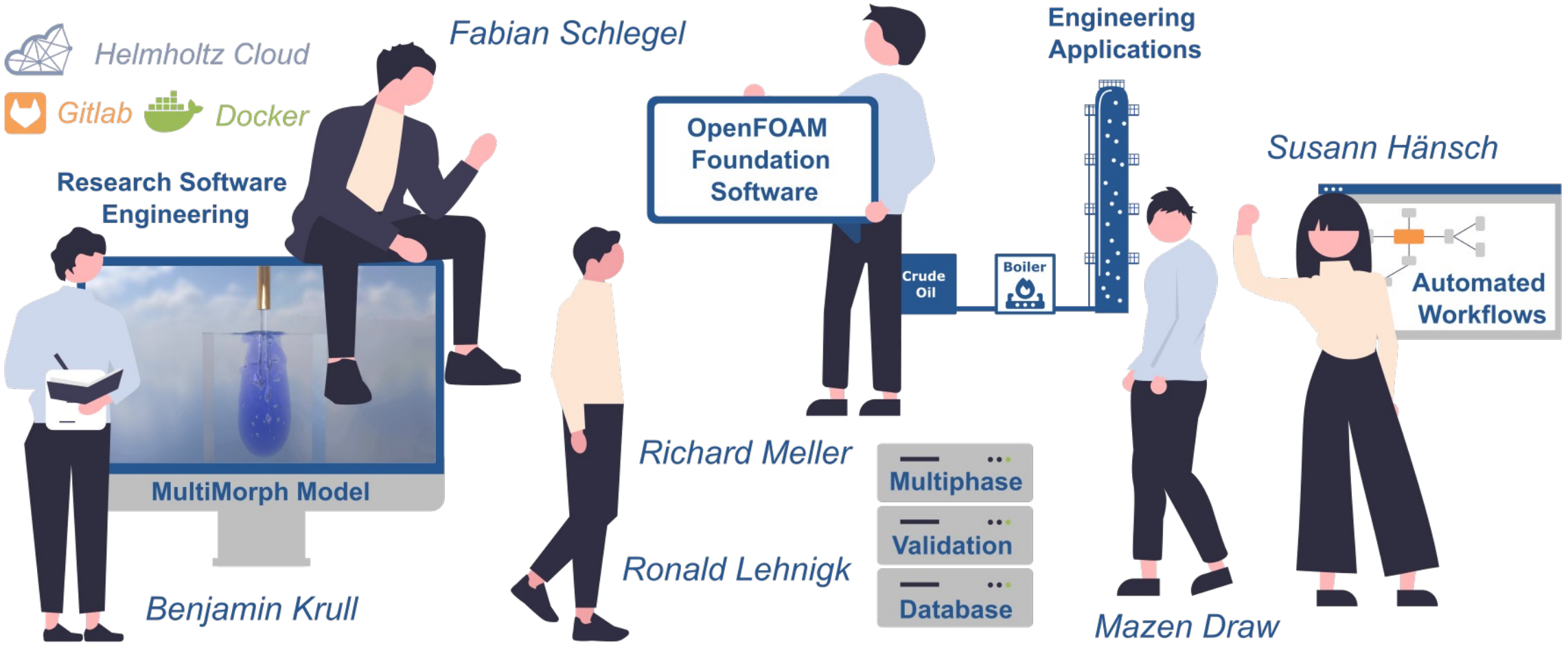
approx. **25 people** including approx. 12 scientists and 8 PhDs



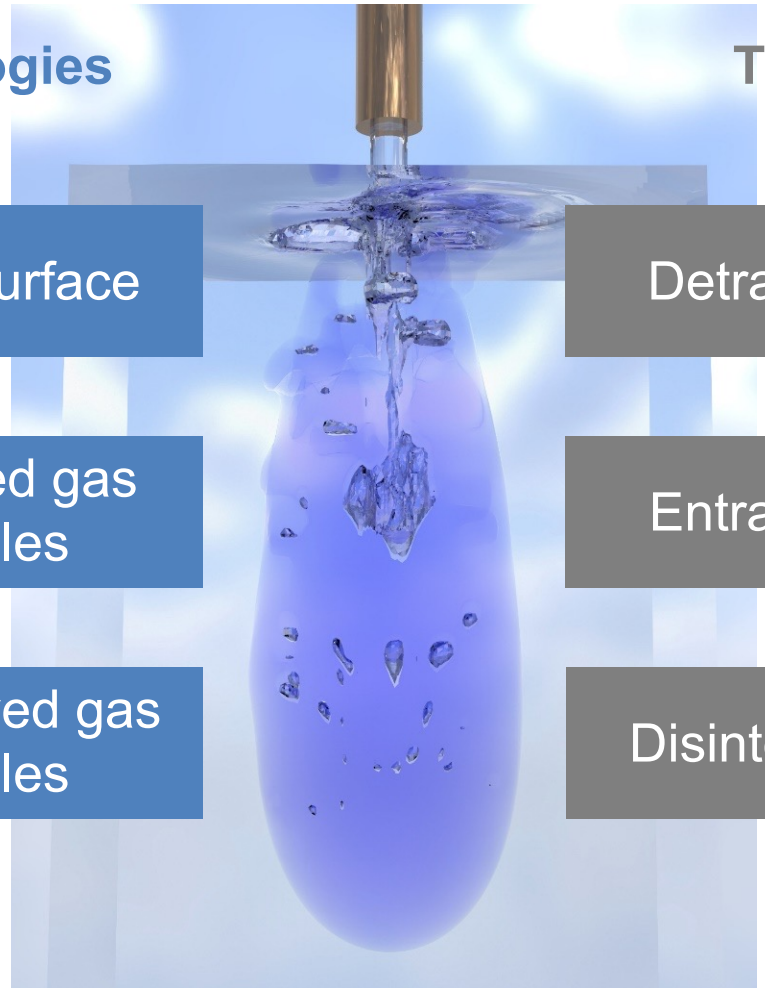
DRESDEN concept HZDR

# The OpenFOAM Group and our Interests

## Numerical simulation of multiphase flows in engineering applications



## Modern simulation methods at HZDR

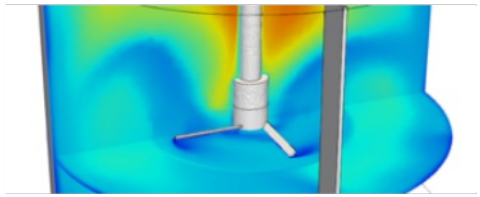



Simulation of a plunging jet with **MultiMorph model**

# WHAT ARE **WE DOING?**

# Scales and Accuracy in numerical simulations

efficiency,
grid spacing



System-scale simulation

Engineering-scale simulation

Reynolds-Averaged Navier-Stokes (RANS)

Large-Eddy simulation (LES)

Direct numerical simulation (DNS)

Molecular-scale simulation

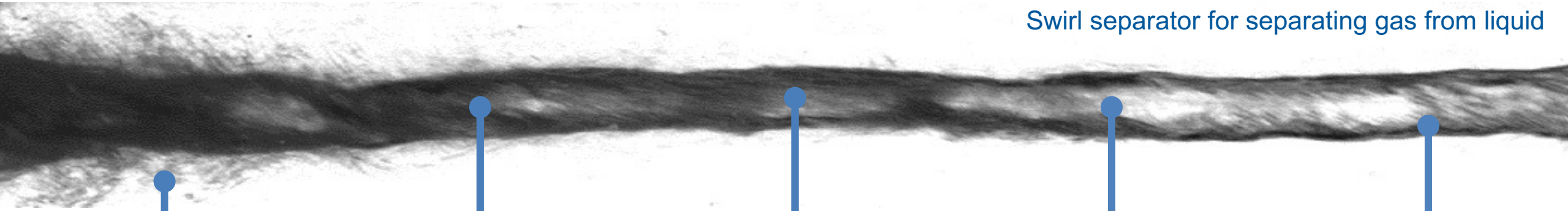
Which method should
we use?


scale, modelling, averaging level



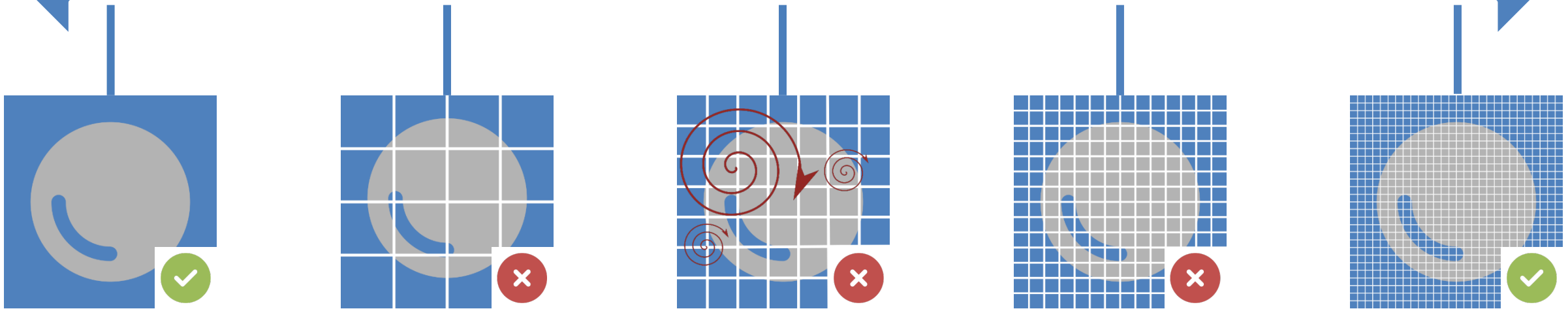
DRESDEN concept HZDR

# MULTIPHASE FLOWS CHARACTERISED BY BROAD RANGE OF SCALES



Swirl separator for separating gas from liquid

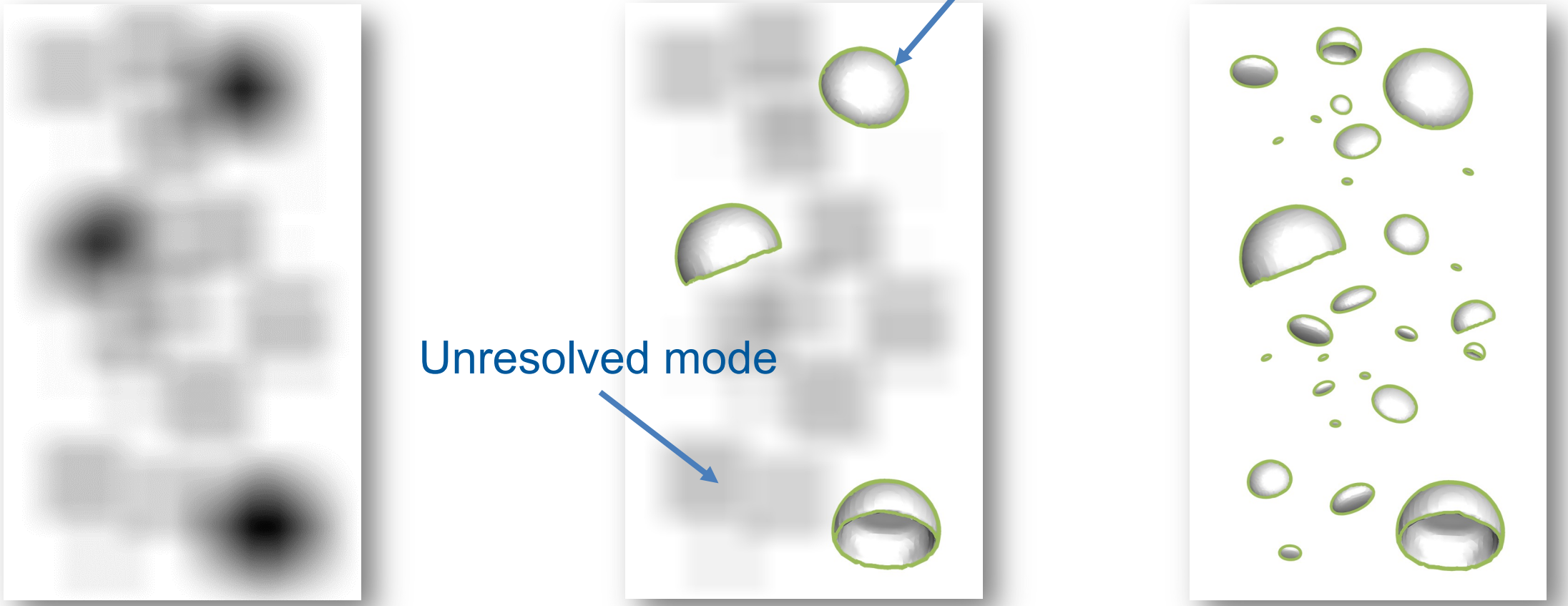
Industrial applications contain **all morphologies**





DRESDEN concept HZDR

# ILLUSTRATION OF INTERFACE TREATMENT




Euler-Euler Model

**MultiMorph**

Volume-of-Fluid



DRESDEN concept HZDR

# Development status of MultiMorph Model

## Resolved interface

Meller et al. (2021). **Basic verification of a numerical framework applied to a morphology adaptive multifield two-fluid model considering bubble motions**. *Int J Num Method Fluid*, 93(3), 748-773.

## Under-resolved bubbles

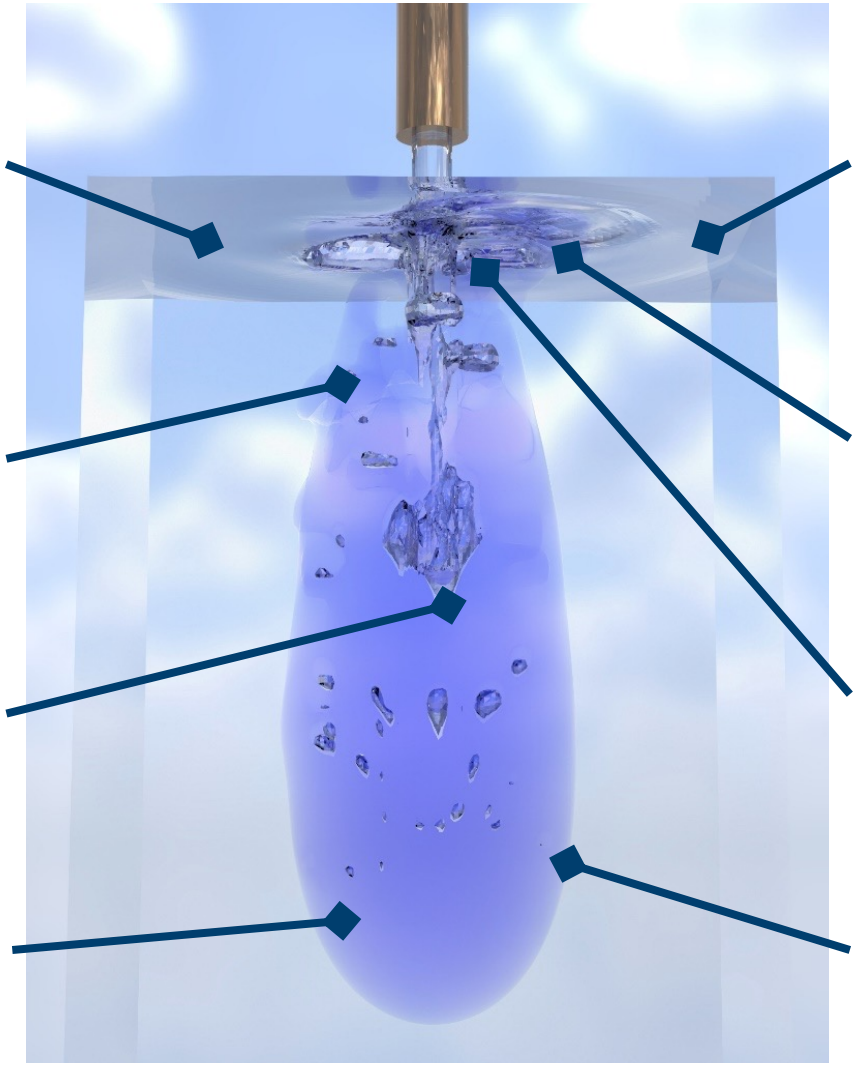
Meller et al. (2023). **Momentum exchange modeling for coarsely resolved interfaces in a multifield two-fluid model.** *Int J Num Method Fluid*, *95*(9), 1521-1545.

## Disintegration

Meller et al. (2024). **Numerical Transfer Towards Unresolved Morphology Representation in the MultiMorph Model.** *Nucl Eng Design*, 428, 113470.

## Over-resolved bubbles

Krull et al. (2024). **A filtering approach for applying the two-fluid model to gas-liquid flows on high resolution grids**. *Chem Eng Sci*, 119909.



## Interface turbulence

Tekavčič et al. (2021). **Validation of a morphology adaptive multi-field two-fluid model considering counter-current stratified flow with interfacial turbulence damping**. *Nucl Eng Design*, *379*, 111223.

## Accumulation

Yin et al. (2023). **A CFD approach for the flow regime transition in a vane-type gas-liquid separator.** *Int J Multiphase Flow*, *159*, 104320.

## Empirical entrainment

Wang et al. (2023). **Simulation of droplet entrainment in annular flow with a morphology adaptive multifield two-fluid model**. *Phys Fluid*, *35*(10).

## General concept

Schlegel et al. (2023). **Openfoam-hybrid: A morphology adaptive multifield two-fluid model.** *Nucl Sci Eng, 197*(10), 2620-2633.

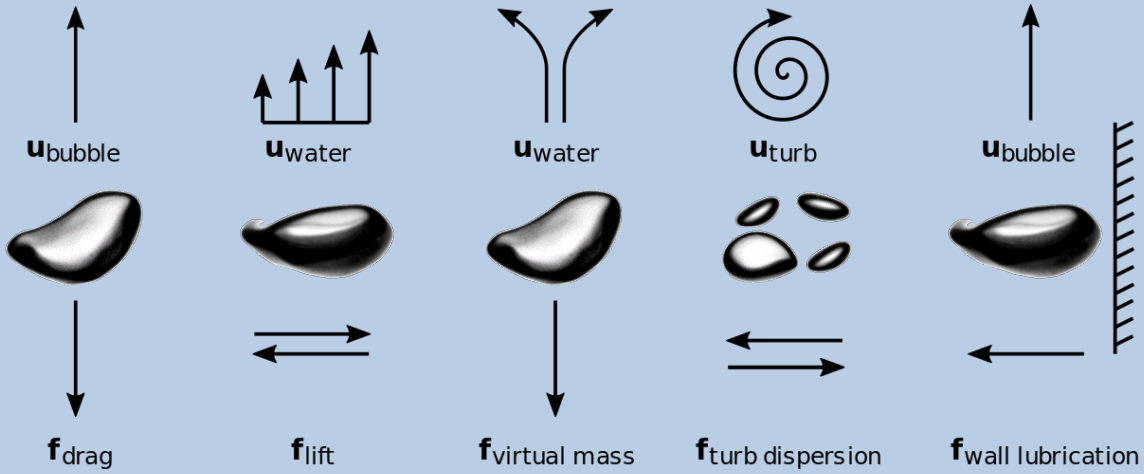


DRESDEN concept HZDR

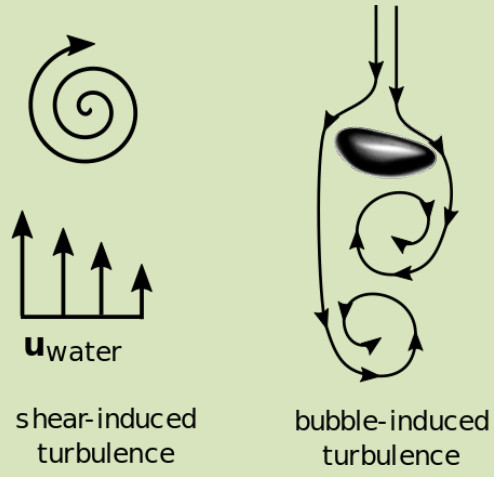
# BASELINE MODEL FOR POLY-DISPERSE GAS LIQUID FLOWS

## CLOSURE MODELS NEEDED FOR EULER-EULER SIMULATIONS

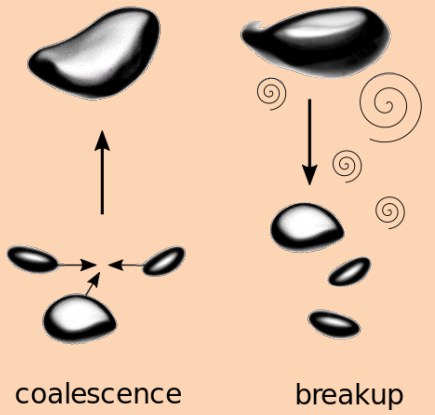
### Interfacial models




### Turbulence

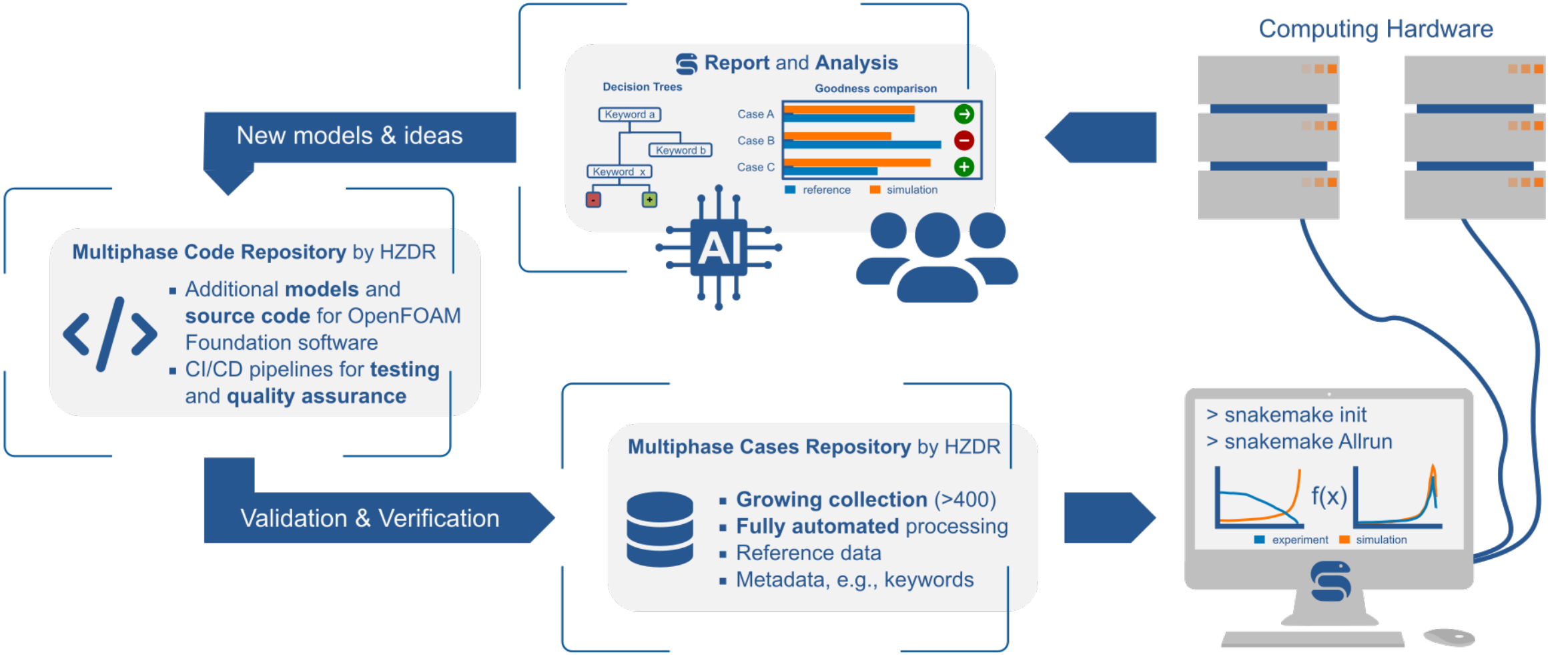



### Bubble interactions




► Lecture on Multiphase Flows planned for 2025

# VALIDATION STRATEGY FOR MULTIPHASE FLOWS



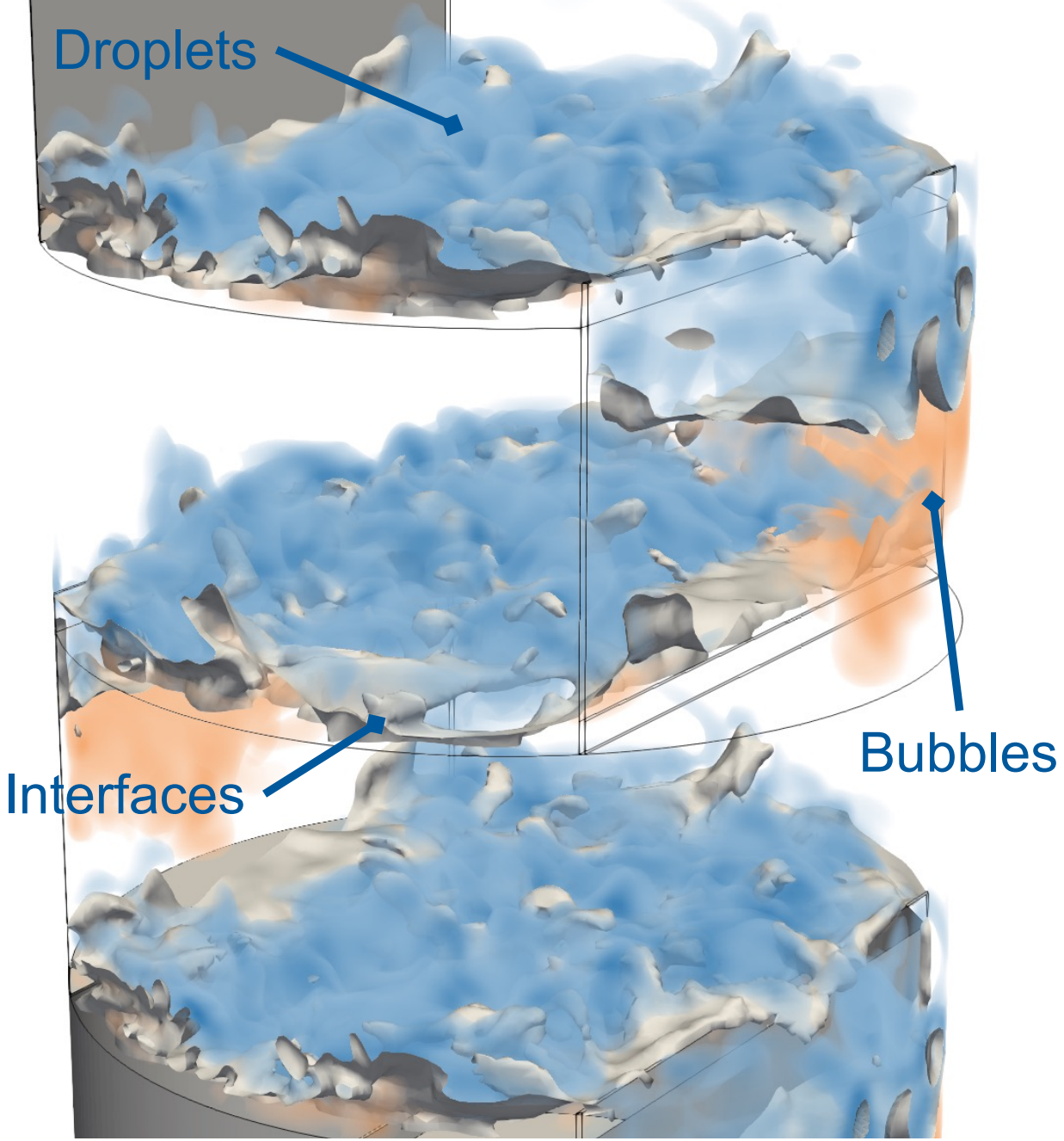
# SOFTWARE PUBLICATIONS

Schlegel et al. (2024, August 22). **Multiphase Code Repository by HZDR for OpenFOAM Foundation Software** (Version 11-s.1-hzdr.1). Rodare. http://doi.org/10.14278/rodare.3105

Hänsch et al. (2024, August 22). **Multiphase Cases Repository by HZDR for OpenFOAM Foundation Software** (Version 11-s.1-hzdr.1-cases.1). Rodare. http://doi.org/10.14278/rodare.3104

Schlegel et al. (2024, October 10). **Multiphase Python Repository by HZDR** (Version 1.3.1). Rodare. http://doi.org/10.14278/rodare.3191

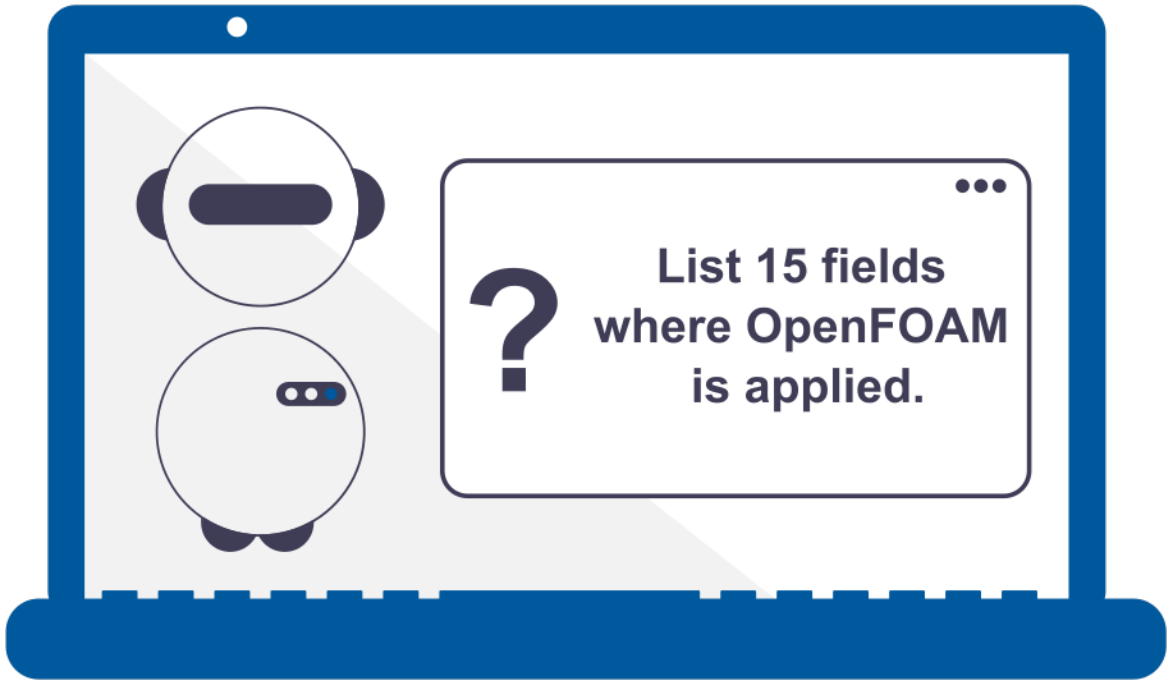
https://hzdr.de/multimorph




Simulated two-phase flow in a tray column

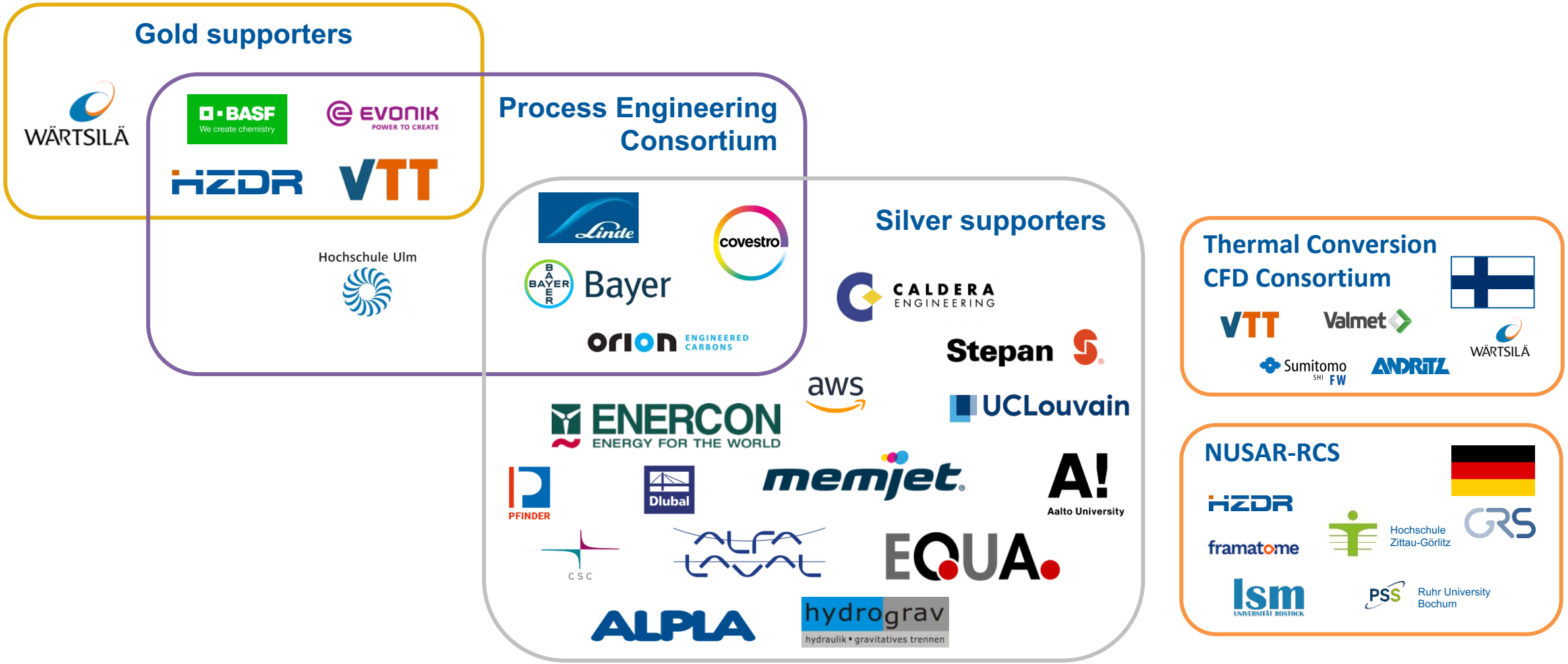
# DIVING INTO OPENFOAM FOUNDATION SOFTWARE

# Field of Applications for OpenFOAM Foundation Software




Generated by ChatGPT

# COMMUNITY FOR OPENFOAM FOUNDATION SOFTWARE





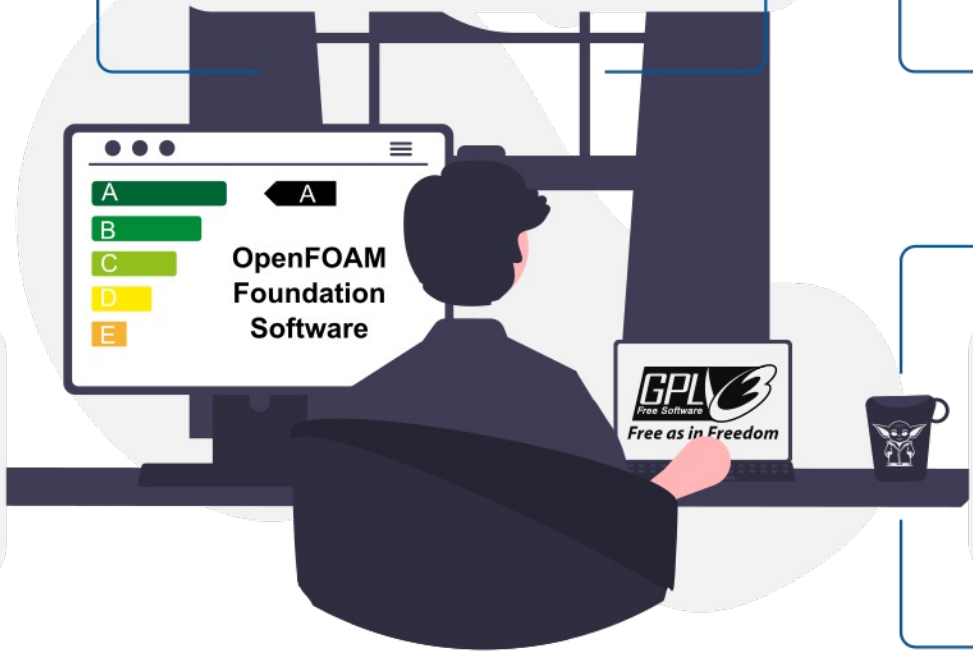

DRESDEN concept HZDR

# THE PACKAGE BY OPENFOAM FOUNDATION SOFTWARE

**Pre-Processing**
- Converters
- Meshing
- Efficient case setup, e.g., *foamGet foamDictionary, foamCloneCase, foamSearch*

**Running / Solving**
- Discretisation schemes
- Matrix solvers
- Pluggable solver modules, e.g. *incompressibleFluid, multiphaseEuler*
- Parallelisation

**Post-Processing**
- functionObjects for analysis, e.g., *foamPostProcess*
- Converters
- Visualisation, e.g., *foamCreateVideo, foamVTKSeries*

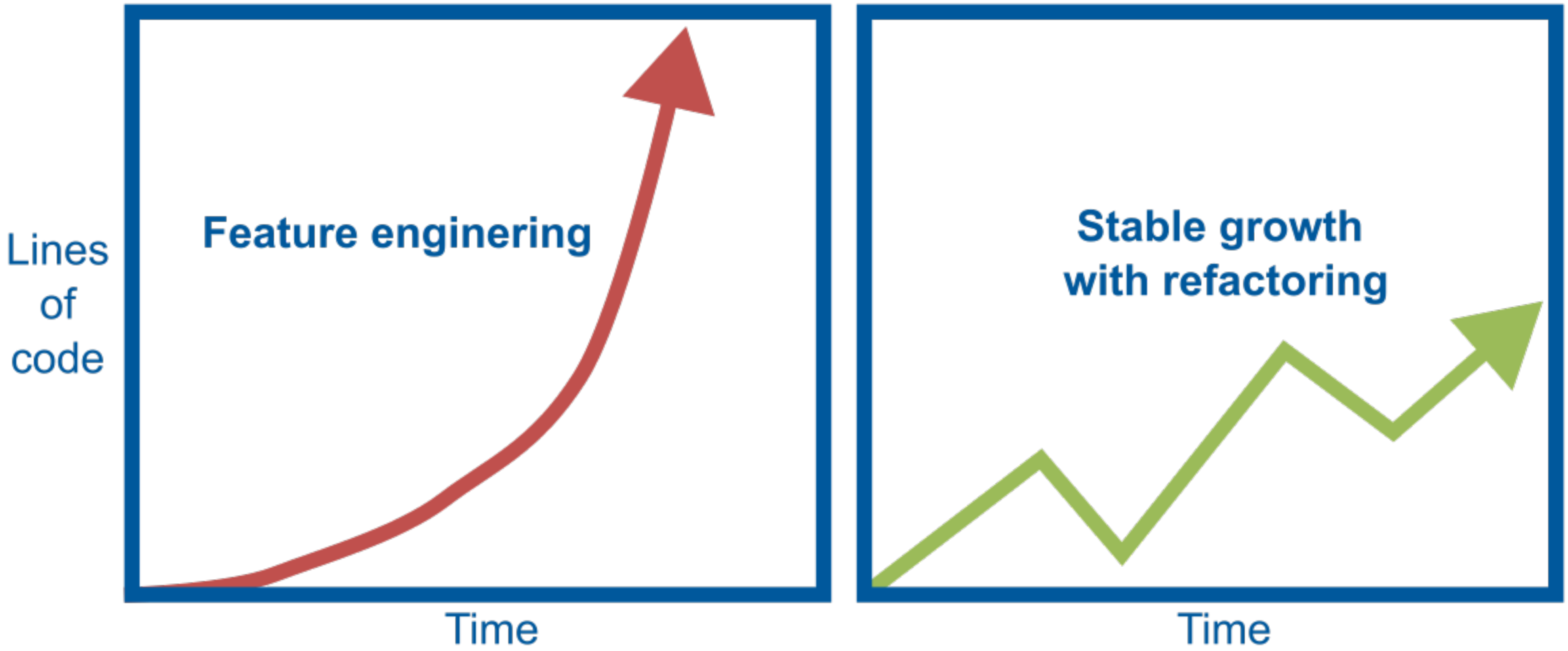



**Documentation**
- Doxygen for source code
- more than 250 tutorials
- various utilties, e.g., *foamFind, foamHelp, foamUnits*
- The Book*

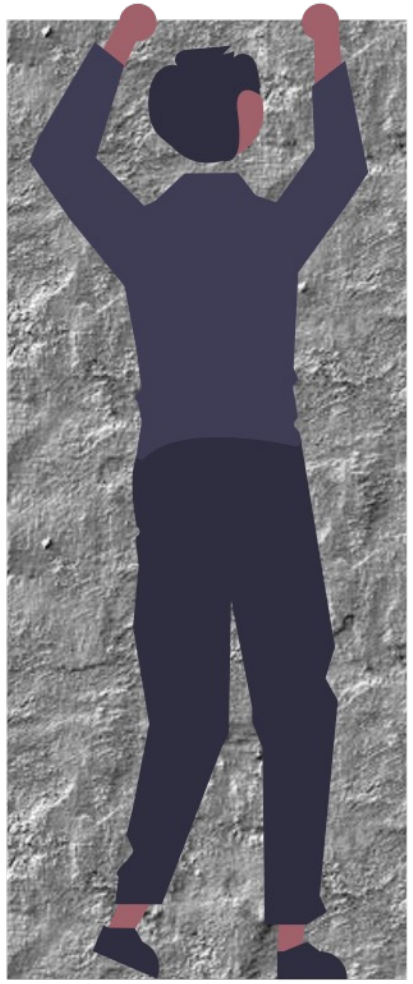
**Customisation**
- 1.5 Mio. lines of source code
- Modular object-oriented design
- Domain-specific language
- Robust and transparent

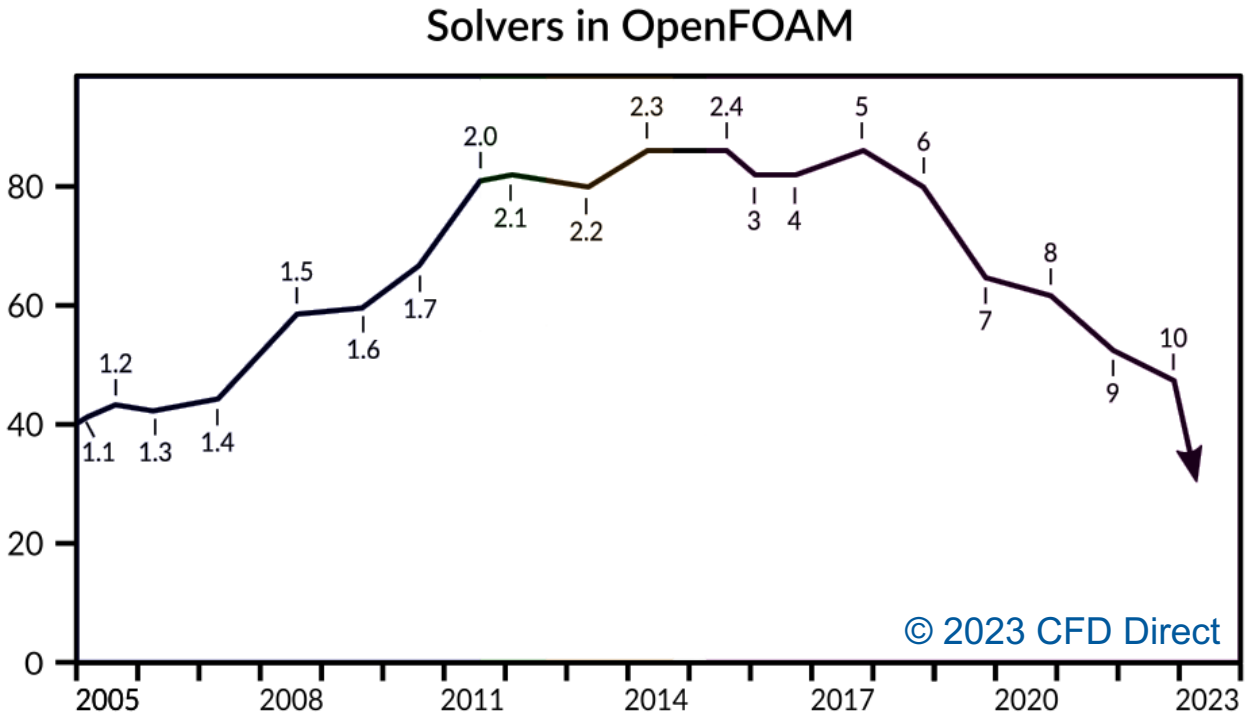
*Greenshields and Weller, Notes on Computational Fluid Dynamics: General Principles, CFD Direct Limited

# Feature engineering vs stable growth



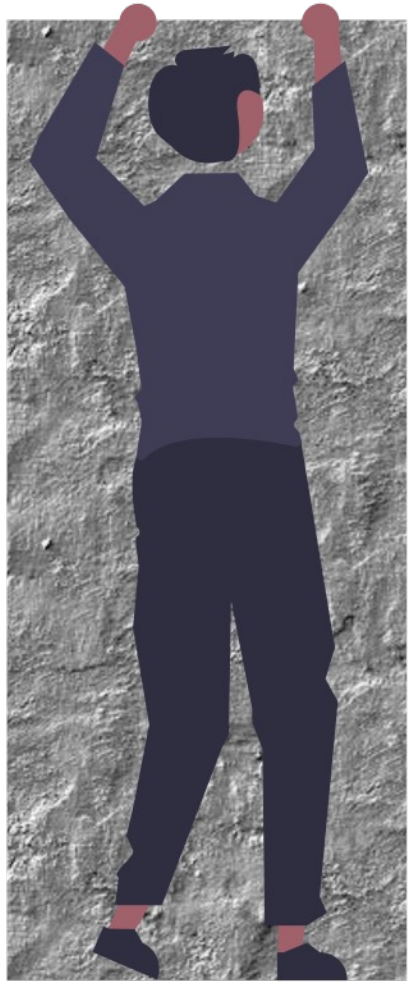
# LOOK ACROSS THE WALL – SOLVER MODULES






- Reduce complexity and need for customisation
- Enabling multi-region simulations, e.g. conjugate heat transfer

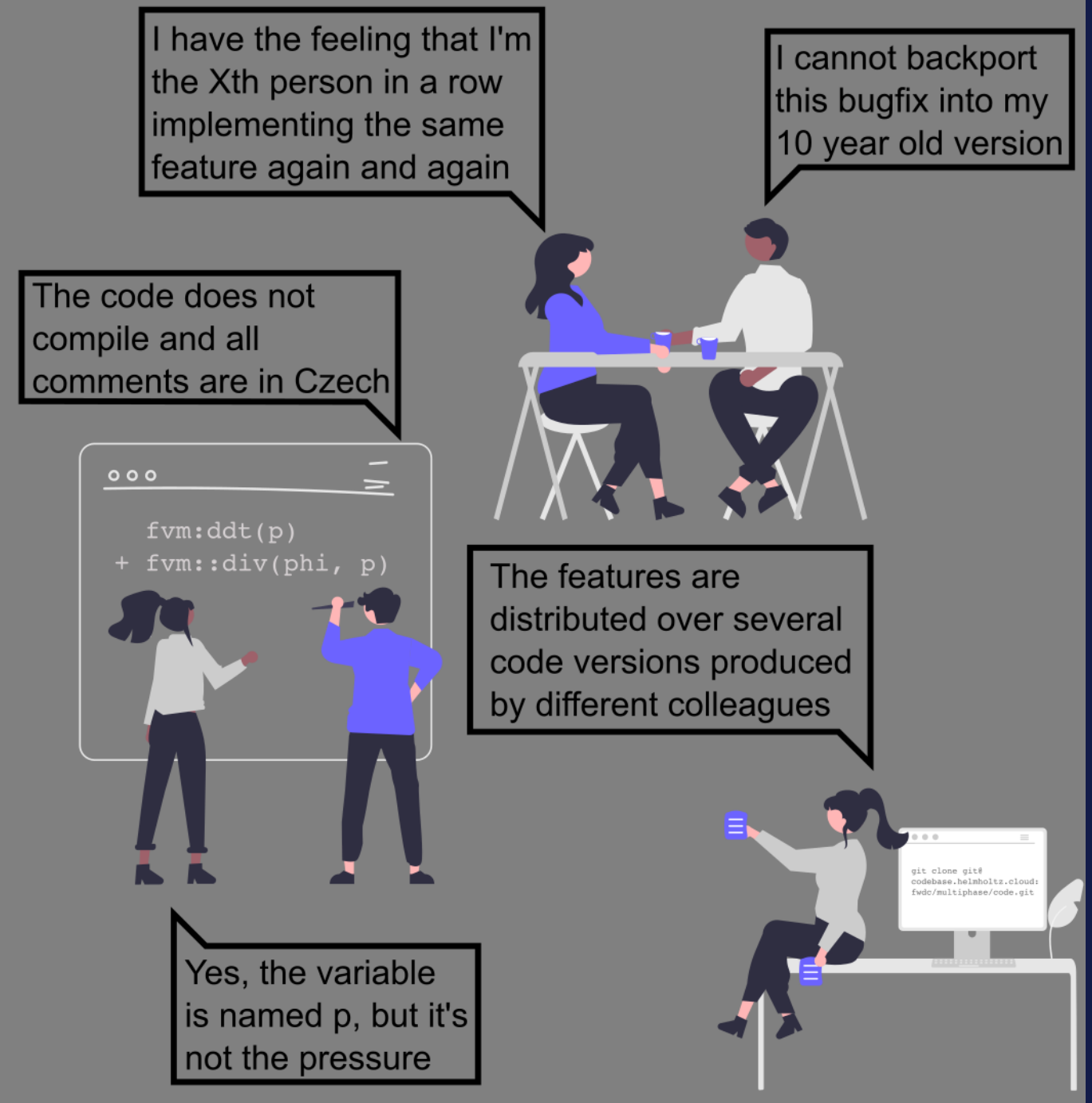
# LOOK ACROSS THE WALL – NEW UNIT SYSTEM



Now possible to specify units for majority of input parameters
(automatically converted to SI units by OpenFOAM Fundation software)

```
boundaryField
{
        inlet
        {
                type flowRateInletVelocity;
                volumetricFlowRate 0.1 [l/s];
                value $internalField;
        }
}
```

Standard: kg, m, s, K, kmol, A, Cd
Derived: Hz, N, Pa, J, W, g, um, mm, cm, km, l, ml, us, ms, min, hr, mol, rpm, bar, atm, kPa, Mpa, cal, kcal, cSt, cP, %, rad, rot, deg



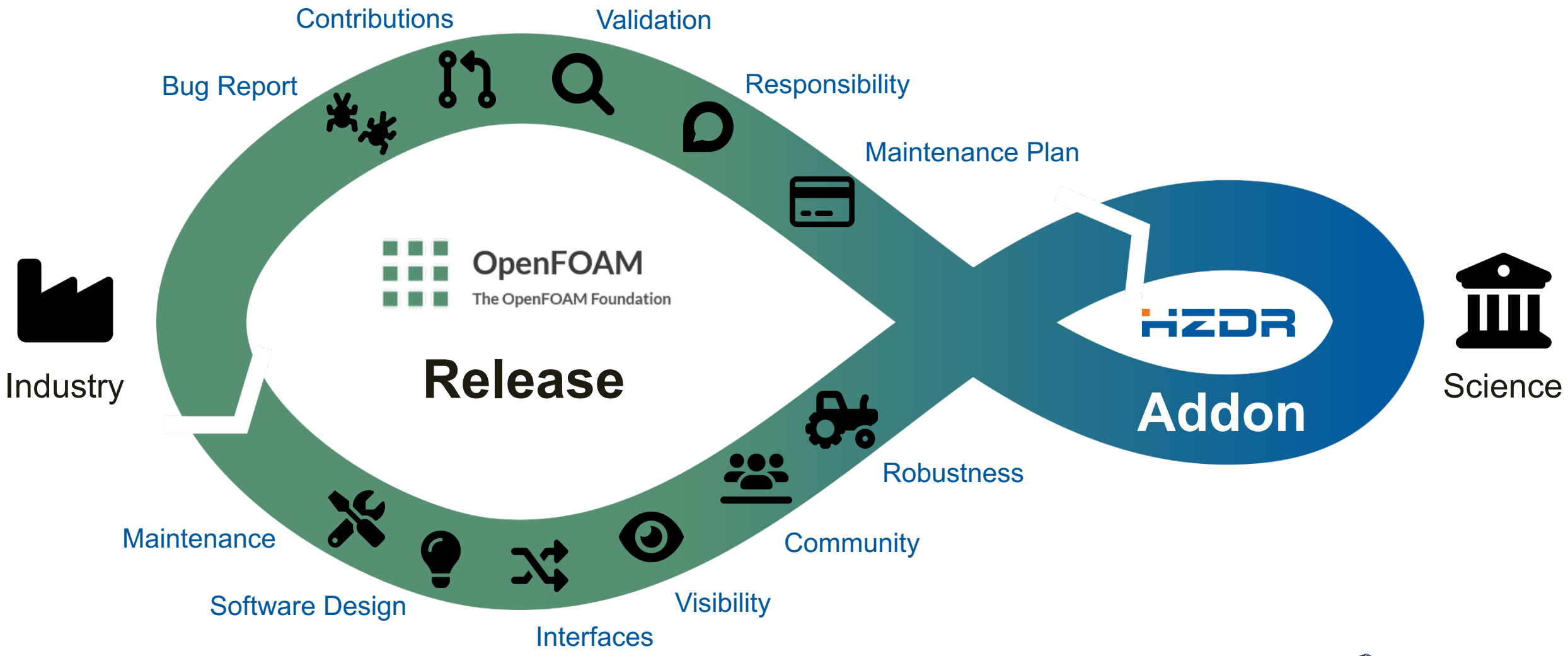

# SUTAINABLE DOWNSTREAM DEVELOPMENT

# Our development philosophy

- Use OpenFOAM Foundation utilities whenever possible
- Provide templates for cases including documentation
- Automate everything possible (Gitlab CI/CD, Snakemake)
- Pre-installed working environment (Docker, VSCode)
- Findable, Accessible, Interoperable, Reproducable (FAIR)
- Adopt standards by OpenFOAM Foundation
- Relationship to core developers (communication, maintenance & project funding)

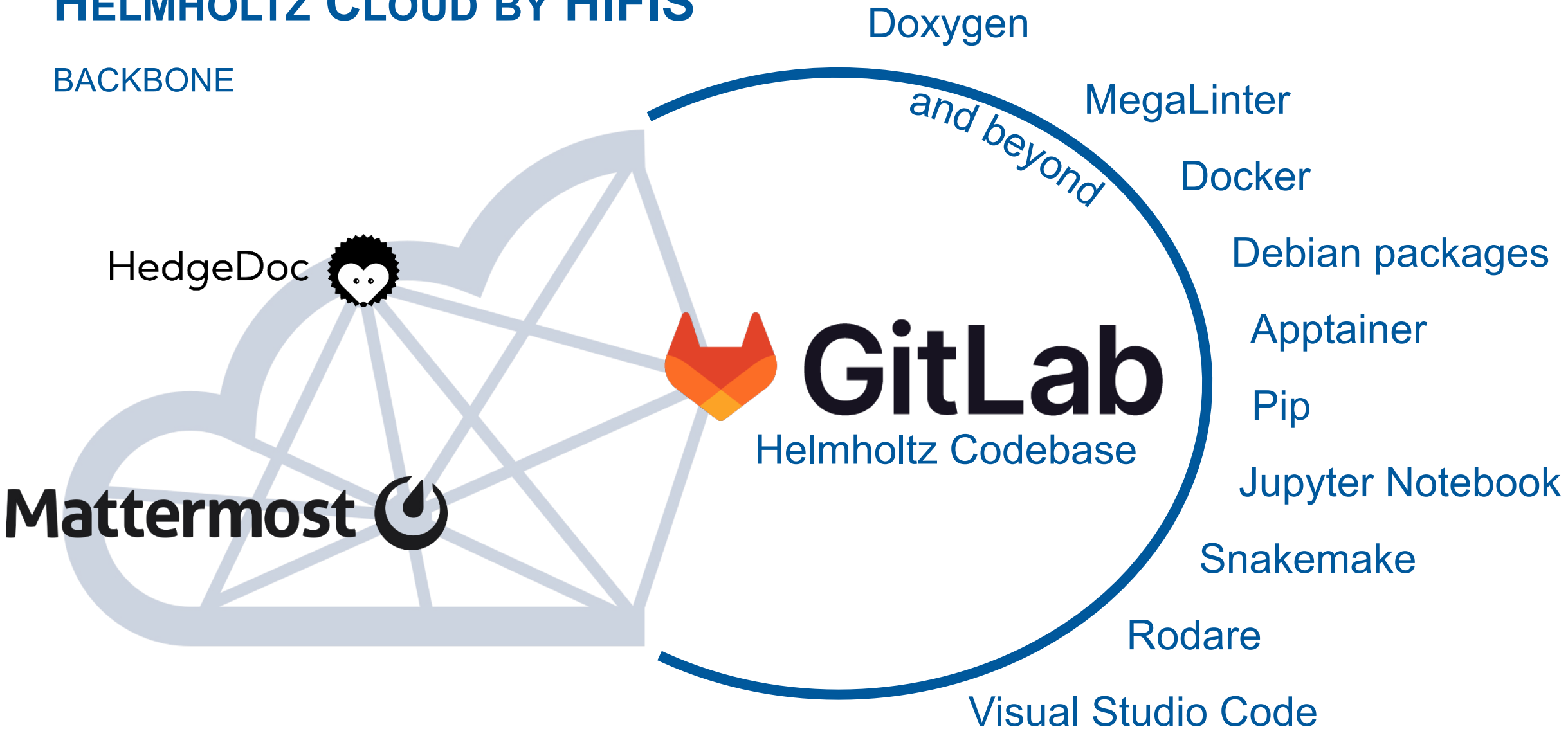
# Contribution to OpenFOAM

## Contributor with addon

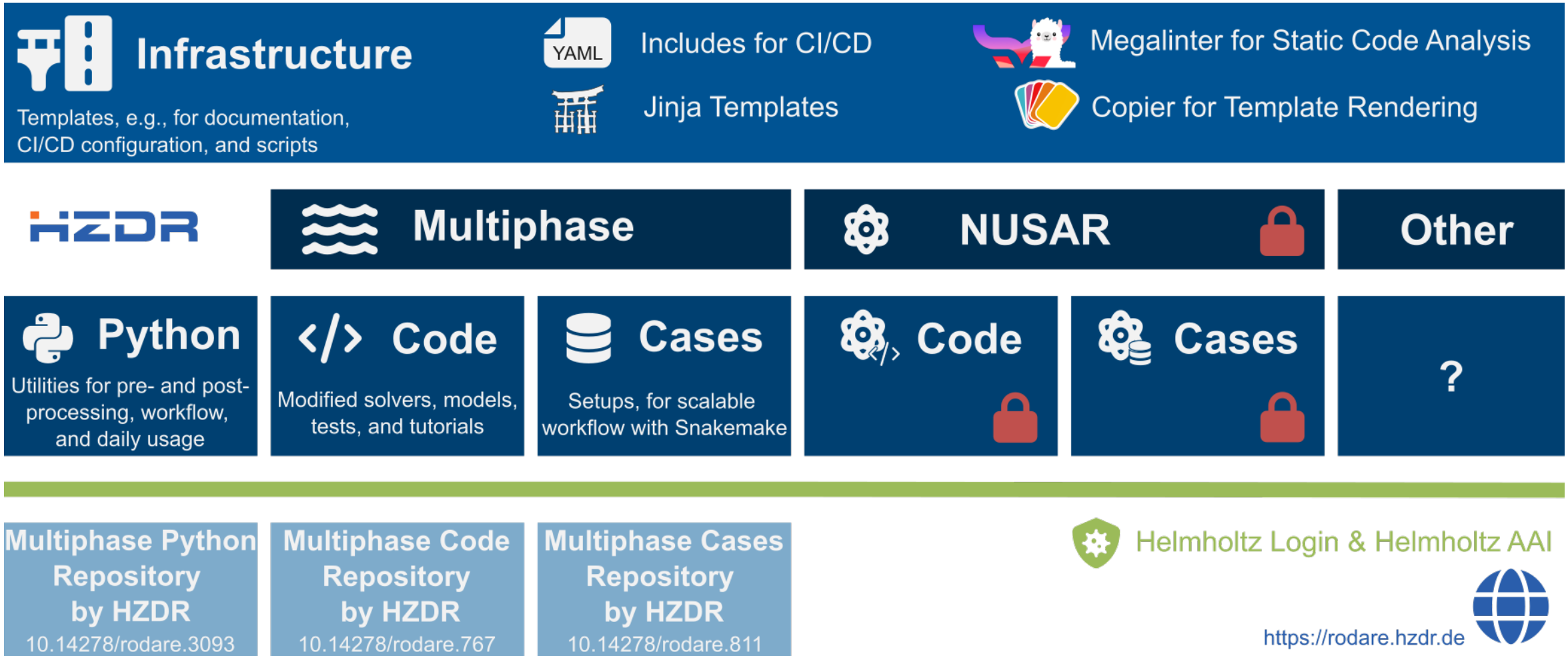


# HELMHOLTZ CLOUD BY HIFIS

BACKBONE



# Our Repositories






DRESDEN concept HZDR

# PORTABLE WORKFLOW WITH GITLAB

**Code**
with Apptainer images

Test runs for downstream project

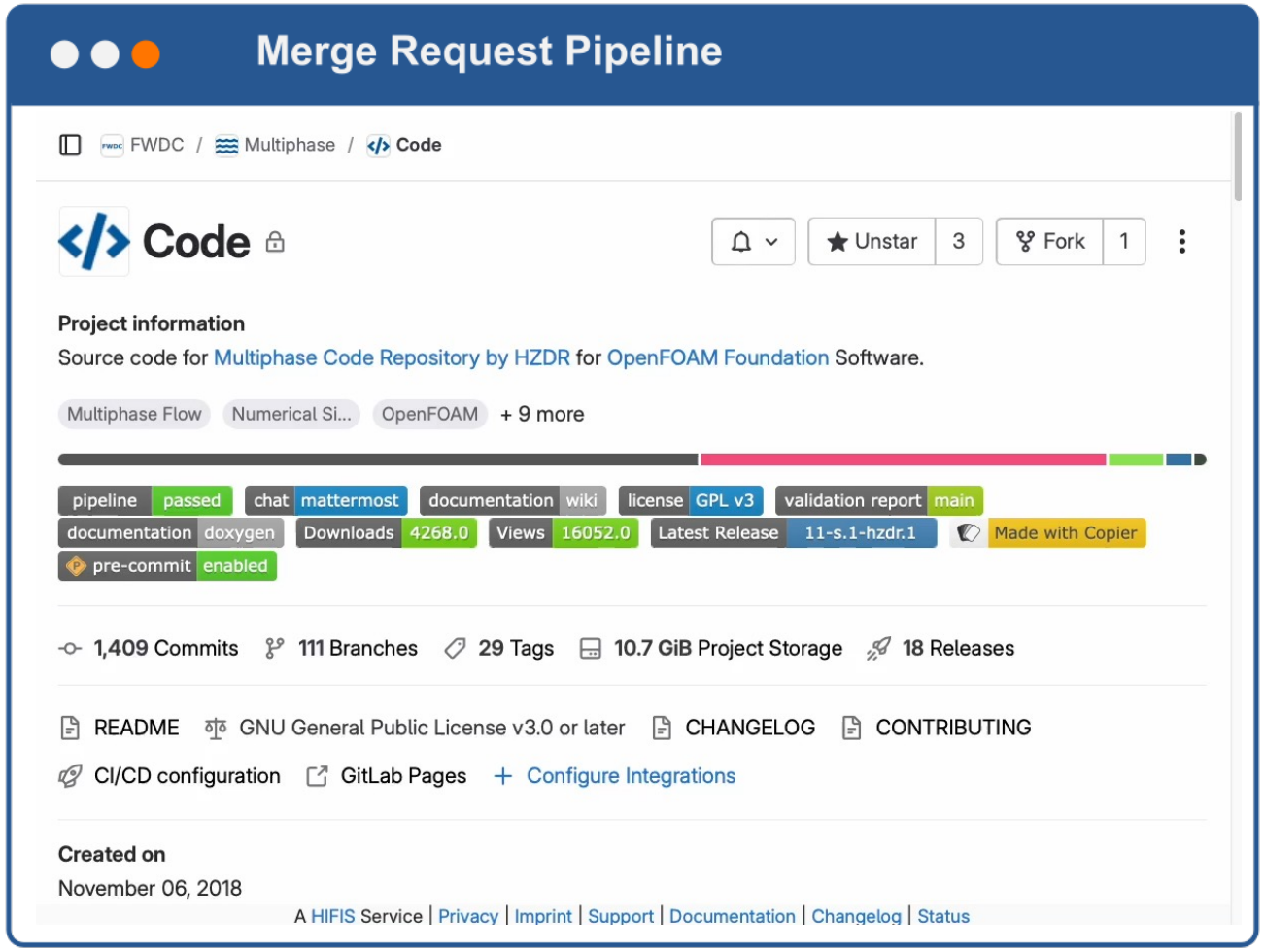
**Cases**
with Snakemake workflow

Scalable execution

**Gitlab Runner**

**Apptainer**
**+**
**Snakemake**
**=**
**Portability**

```
apptainer pull docker://<image>
```



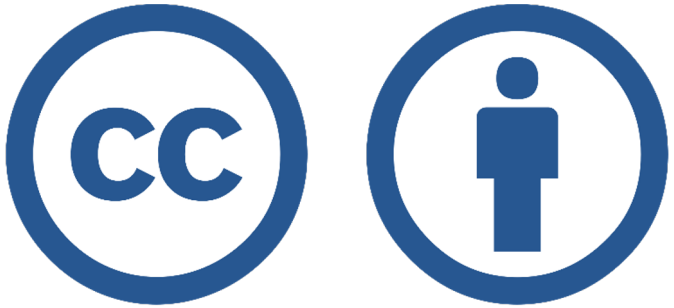
# SUMMARY

- Development of a morphology-adaptive Euler-Euler method for engineering scale simulations
- Validation strategy for numerical simulations of multiphase flows
- OpenFOAM Foundation software is the perfect package for us
- Applying modern IT-tools and CI/CD strategies to allow sustainable research

► Demo on selected tools to organize simulation setups by Ronald Lehnigk

# ACKNOWLEDGEMENT

## LICENSE



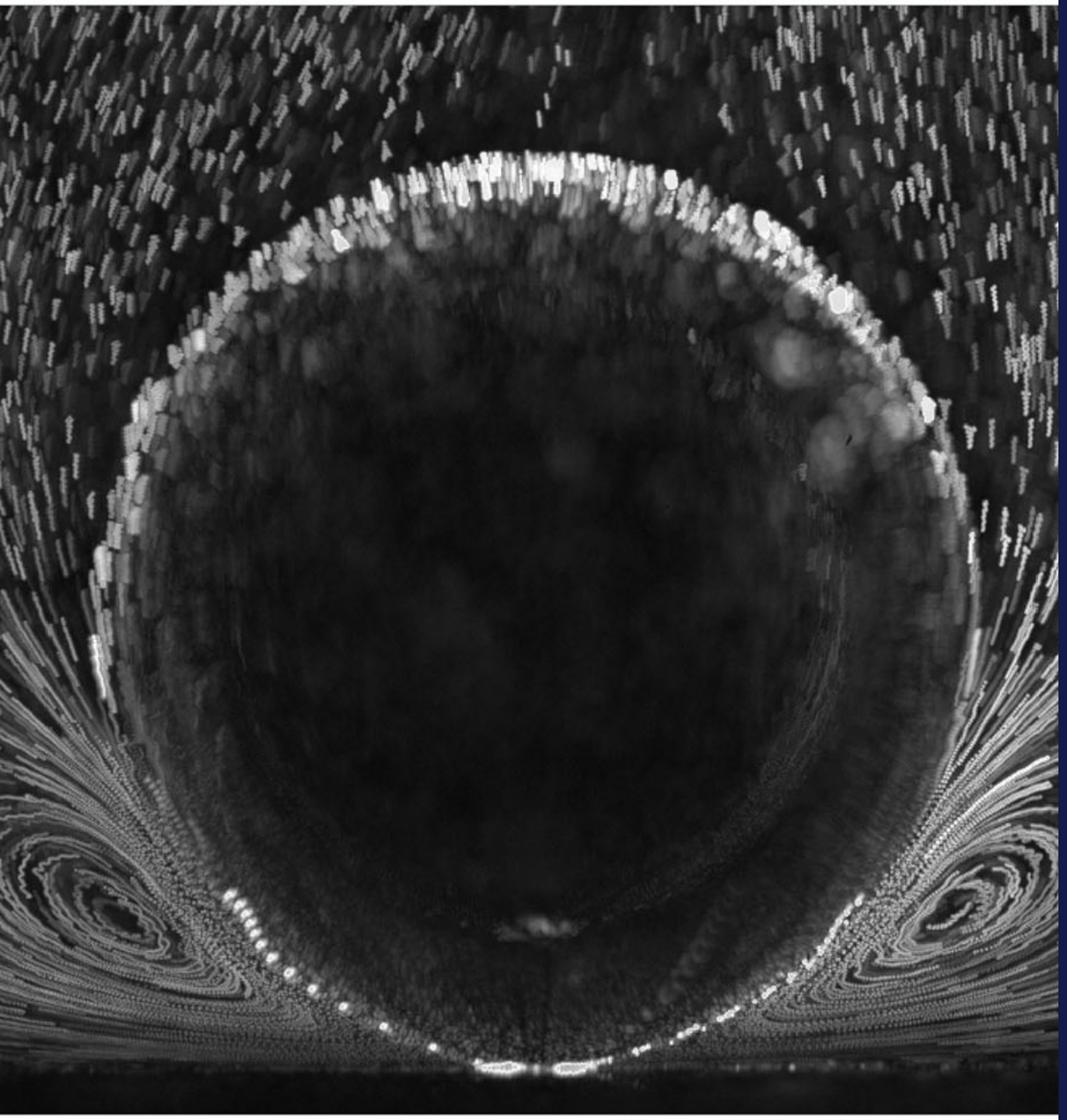
**Creative Commons Attribution 4.0 International**

**You are free to:**

**Share** - copy and redistribute the material in any medium or format
**Adapt** - remix, transform, and build upon the material for any purpose, even commercially.

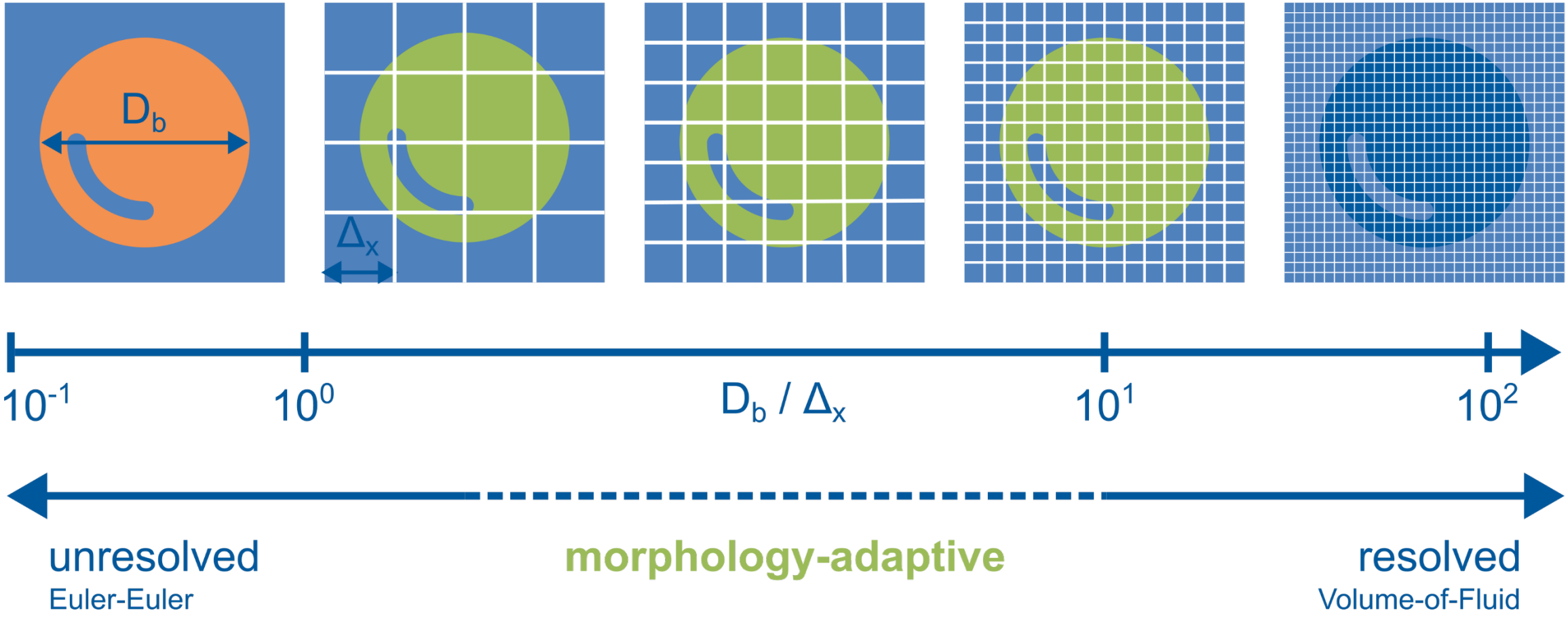
**Under the following terms:**

**Attribution** - You must give appropriate credit, provide a link to the license, and indicate if changes were made.
You may do so in any reasonable manner, but not in any way that suggests the licensor endorses you or your use.



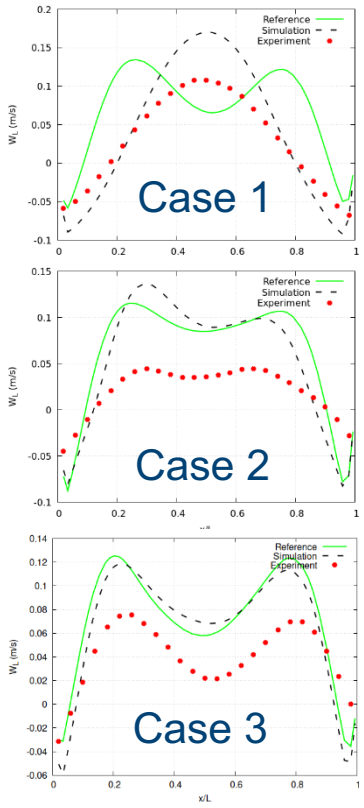
# Slides as **Backup**

# A Morphology-Adaptive Multifield Two-Fluid Model



# QUANTIFYING THE CFD PREDICTION QUALITY

Do results improve with a new model/setup/code?




Metric 1

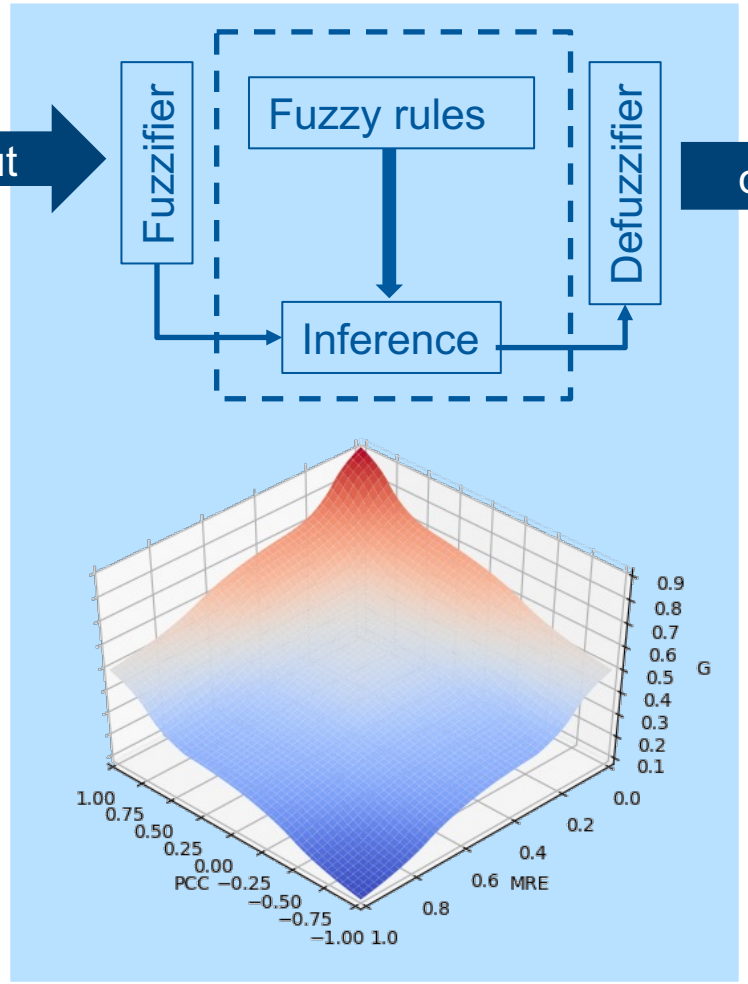
Metric 2

…

extract

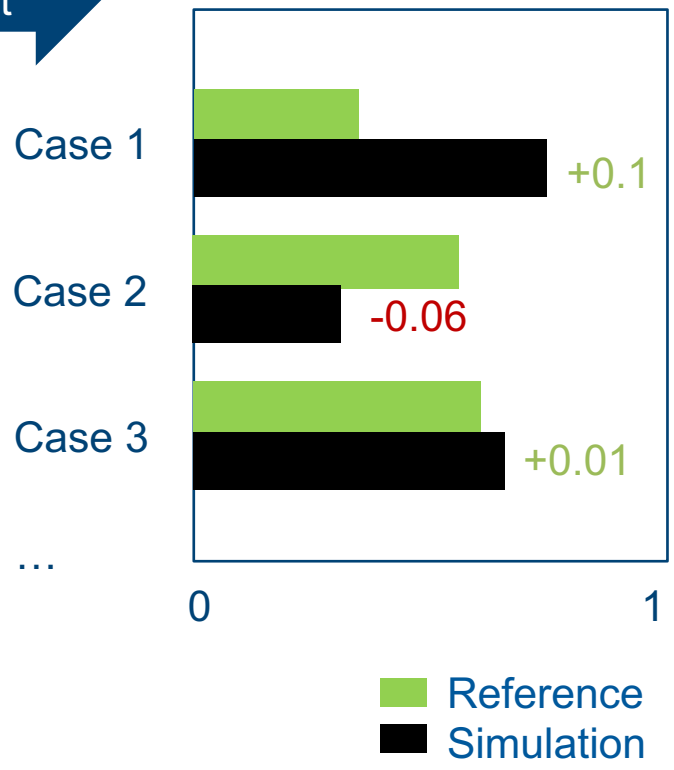
input

Fuzzy logic controller



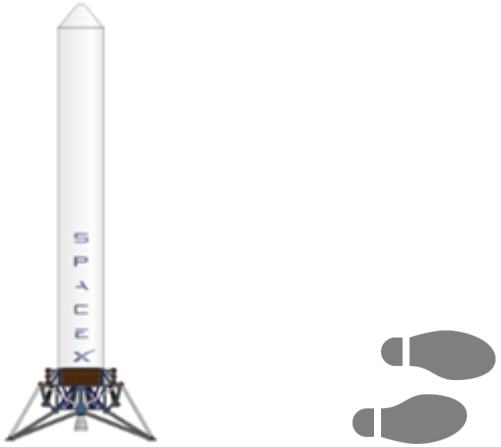

output

Prediction quality



# Quality of Software Engineering

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



Grasshopper test vehicle

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

Falcon 9R Test Vehicle

Falcon 9 FT

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

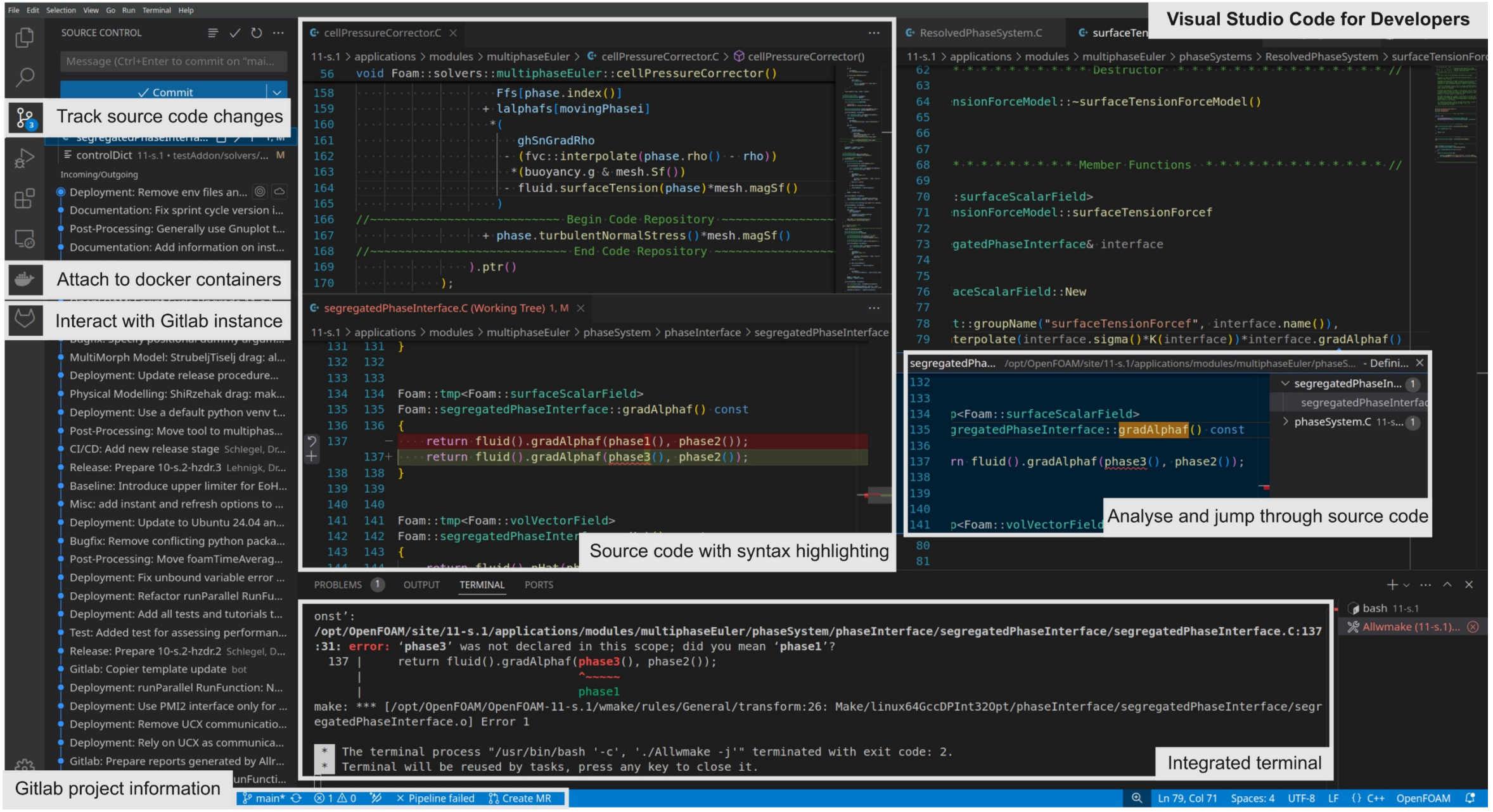
Core Maintainers

**Production Software**
- All features
- Fully optimized
- Production ready



DRESDEN concept HZDR

# Visual Studio Code for Developers



# Visual Studio Code for Users