

Retrospect, Status and Perspective of the Development of containmentFOAM

S. Kelm, M. Kampili, G. Vijaya Kumar, X. Liu, A. George, R. Ji,
L.M.F. Cammiade, K. Arul Prakash, H.-J. Allelein

IAEA Technical Meeting on the Development and Application of Open-Source Modelling and Simulation Tools for Nuclear Reactors, Politecnico di Milano, Italy, June 20-24th 2022

containment  FOAM

Supported by:




Federal Ministry
for the Environment, Nature Conservation,
Nuclear Safety and Consumer Protection


based on a decision of
the German Bundestag
Project No. 150 1633B

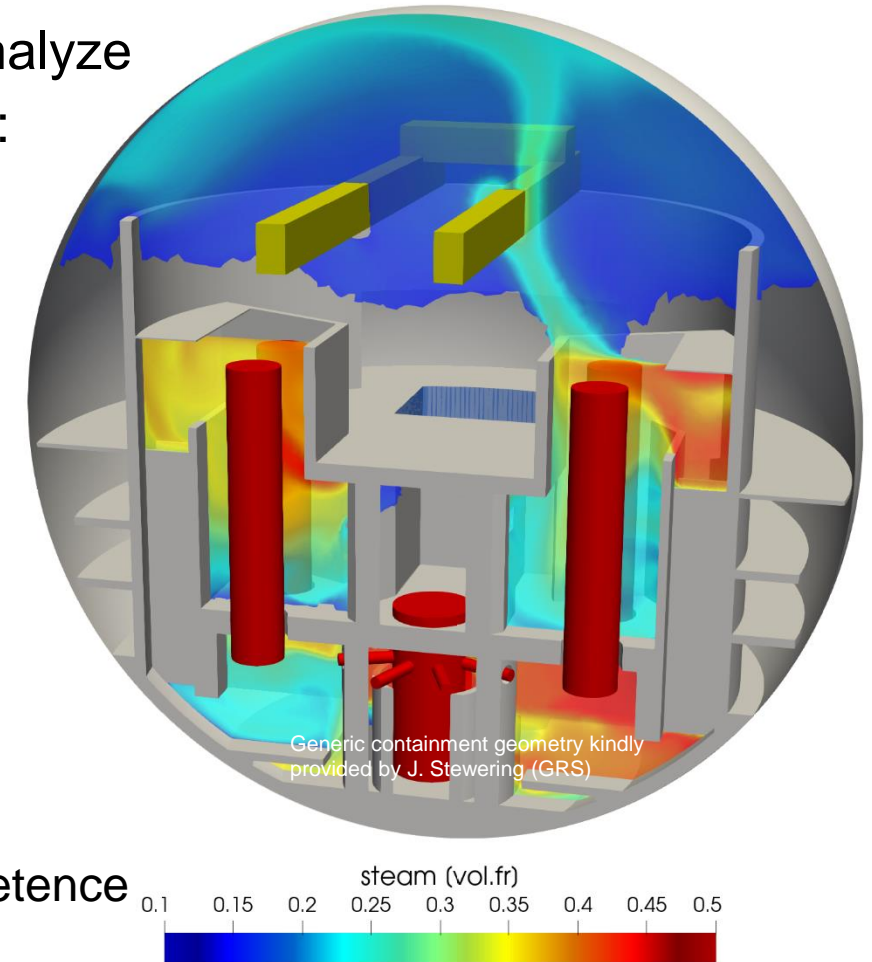
 **JÜLICH**
Forschungszentrum

Mitglied der Helmholtz-Gemeinschaft

- Introduction
 - Background and Motivation
 - Strategy and General Considerations
 - Status of containment  F4AM
- Open-source challenges
 - Maintenance, distribution
 - User-Developer interaction
- Best practices
- Summary and Perspective

BACKGROUND

- Development of containment  [1] as an open basis to analyze containment pressurization, flows, H₂/CO and aerosol behavior:
 - Basis for our research within the Helmholtz Nusafe program:
 - Replacement of previous commercial code basis & legacy code
 - Support experiments & transfer of experimental results to plant scale
 - Investigate interaction of physical phenomena and safety systems under representative conditions
 - Assess effectiveness of (passive) safety systems and measures
 - Contribution to national CFD reference package for NRS
 - Carrier to disseminate our research and ensure its application
 - International cooperation (e.g. IITM, IAEA-ONCORE, OECD/NEA)
 - Attractive platform for education, training and maintenance of competence



[1] Kelm, S. et al. "The Tailored CFD Package 'containmentFOAM' for Analysis of Containment Atmosphere Mixing, H₂/CO Mitigation and Aerosol Transport" *Fluids* (2021) 6, no. 3: 100. <https://doi.org/10.3390/fluids6030100>

CONTAINMENTFOAM ENVIRONMENT

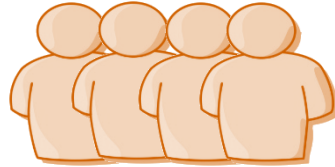
- Active developers team



PI
(permanent)



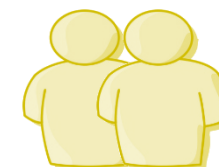
PostDoc



PhD students



students & trainees



technical staff
(permanent)



former
PhD students

➤ Mostly temporary contributors with different levels of qualification!

- Current collaborators, users & beta-testers:



global research for safety



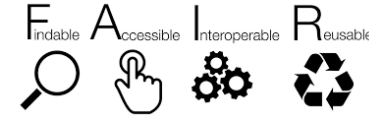
UNIVERSIDAD
POLITÉCNICA
DE MADRID



MOTIVATION TO USE THE OS APPROACH

- Obviously:

- FAIR is the way to do meaningful publicly funded research!



- Many further benefits:

- Visibility of the individual's work and the project:
 - Motivation for contributors and new applicants to join
- Reusability of the research output by others:
 - Increases impact and KPI (citations)
 - In-line with Helmholtz digitalization strategy
- Involvement of temporary staff at different education levels:
 - Possibility to keep them involved beyond their contract
- International collaboration:
 - No troubles with license agreements but maybe contributors agreements needed

DEVELOPMENT OF CONTAINMENTFOAM

Strategy and General Considerations

- Coordinated R&D effort (currently 14 active contributors, cumulative > 25 person years R&D)
 - Multi-scale and Multi-physics application:
 - All physical phenomena and their interaction need to be considered to be representative of an accident progression
 - No separate effect consideration possible, models have to be robustly coupled
 - Model basis has to be well balanced in terms of accuracy and efficiency
 - Baseline set of models
 - Model set with known limitations rather than optimal model for a specific condition
 - User guidance for consistent application of the baseline model
 - Limit maintenance effort
 - Framework / quality assurance
 - Guided case setup and solution monitoring
 - Common post processing (functionObjects), data handling and minimum I/O
 - Software framework for uncertainty quantification



STATUS OF CONTAINMENTFOAM

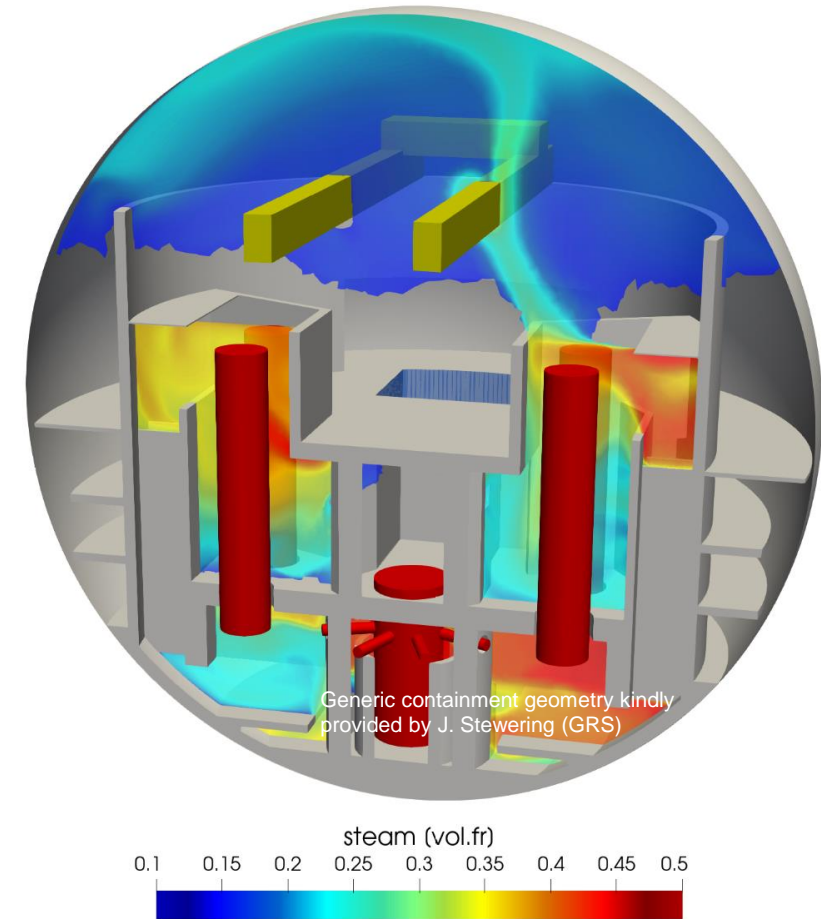
Modeling Pressurization, Aerosol Transport and H₂/CO Gas Mixing and Mitigation

■ Flows and Transport Phenomena

- **Efficient Multi-Species Solver:** effective binary diffusion; Wilke mixture ✓
- **Turbulence transport:** k- ω SST model with buoyancy terms, ✓
simple and generalized gradient diffusion hypothesis formulation ✓
- **Conjugate heat transfer** ✓
- **Wall condensation:** single phase diffusion layer model, ✓
implemented as face fluxes, ✓ dedicated wall treatment ✓
- **Fog formation:** drift flux model with PBM ✓, two phase formulation ✓
- **Gas radiation:** Emission-based Reciprocity Monte Carlo Method, ✓
SNBCK and LBL spectral models ✓
- **Aerosol transport:** LPT with Continuous Random Walk model for turbulent dispersion, ✓ Eulerian drift flux model including hygroscopic growth ✓

■ Technical Systems and Components

- **PARs:** Code coupling with mechanistic model *REKODIREKT* ✓
- **Burst discs, flaps, doors:** conditional mesh interfaces ✓
- **Heat exchangers:** porous media ✓



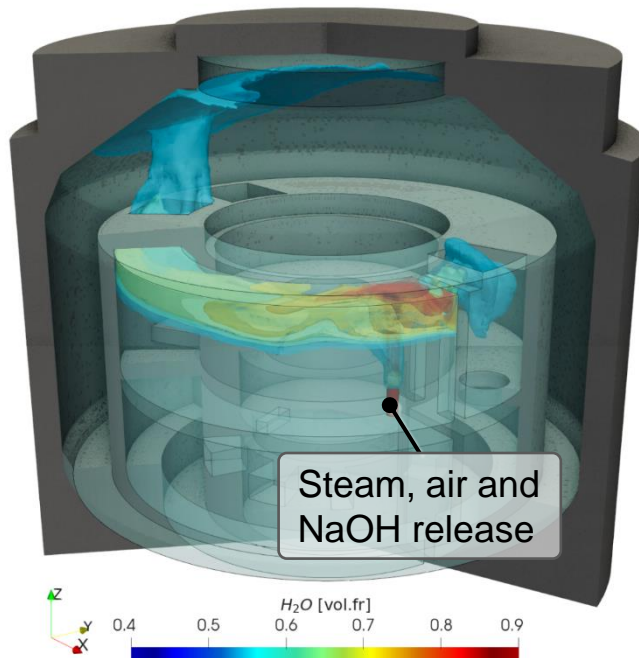
[1] Kelm, S. et al. "The Tailored CFD Package 'containmentFOAM' for Analysis of Containment Atmosphere Mixing, H₂/CO Mitigation and Aerosol Transport" *Fluids* (2021) 6, no. 3: 100. <https://doi.org/10.3390/fluids6030100>

STATUS OF CONTAINMENTFOAM

Applications

■ ISP-37 VANAM M3

– thermo-fluid dynamics and aerosol distribution



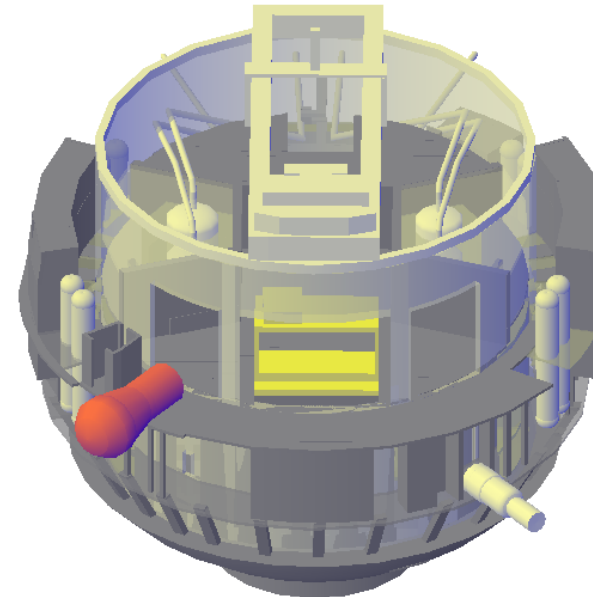
- 600 m³, 20.000 s
- ~ 400 ..1100 s / day @ 128 CPU

■ S. Kelm et al., „Technical Scale CFD Analysis of the Pressurization and Transport Processes in the Battelle Model Containment during the ISP 37 VANAM M3 Test“, accepted for publication at NUTHOS 13, Taichung, Taiwan, September 5-10, 2022

■ L. S. Lopez et al., *Development of a detailed 3D CAD model of a generic PWR-KWU containment as a basis for a better assessment of H₂/CO combustion risk*, Proc. ENYGF'21


■ German KONVOI PWR (1400 MW_{el})

– flammable gas distribution, combustion risk and mitigation





- Work-in-Progress (CAD model by UPM)
- ~ 70000 m³, $O \sim 1000$ s

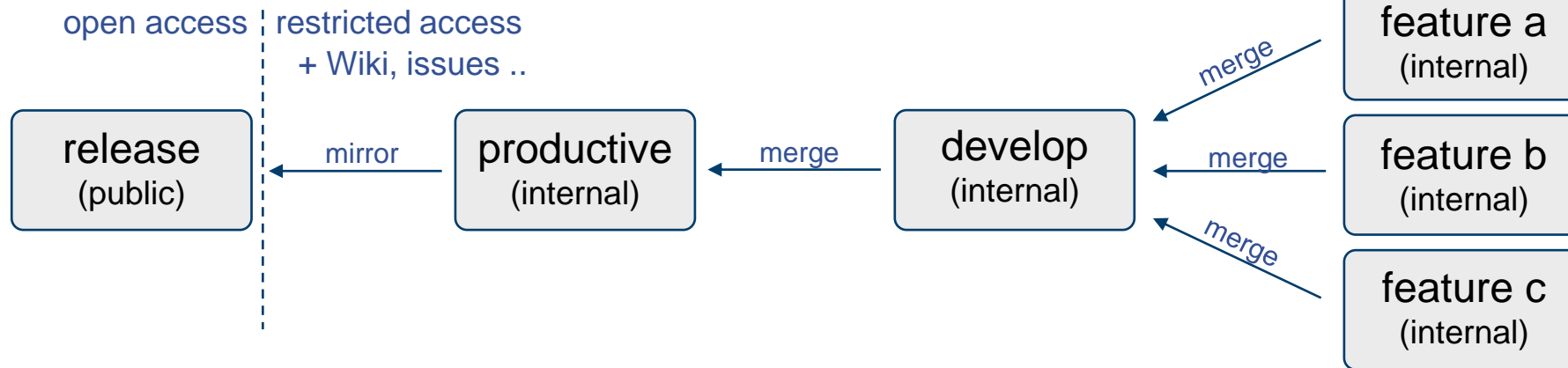


- Introduction
 - Background and Motivation
 - Strategy and General Considerations
 - Status of containment  VFAAM
- Open-source challenges
 - Maintenance, distribution
 - User-Developer interaction
- Best practices
- Summary and Perspective

UNIQUE OPEN SOURCE ISSUES

- Licensing: derived from the original code basis  OpenFOAM : GNU public license v 3.0
- Distribution: via gitlab <https://go.fzj.de/containmentFOAM> 

- Branching model:



- long-term maintenance (supported by BMUV, Project No. 150 1633B in collaboration with GRS, Project No. RS1603A)
 - mostly a 'non-scientific' work (implications on funding or visible research output)
 - but requires skilled scientists who know the models and their numerical implementation peculiarities
 - comprehensive efforts to follow refactoring of of the original code basis (OpenFOAM)


Developer - User Interaction

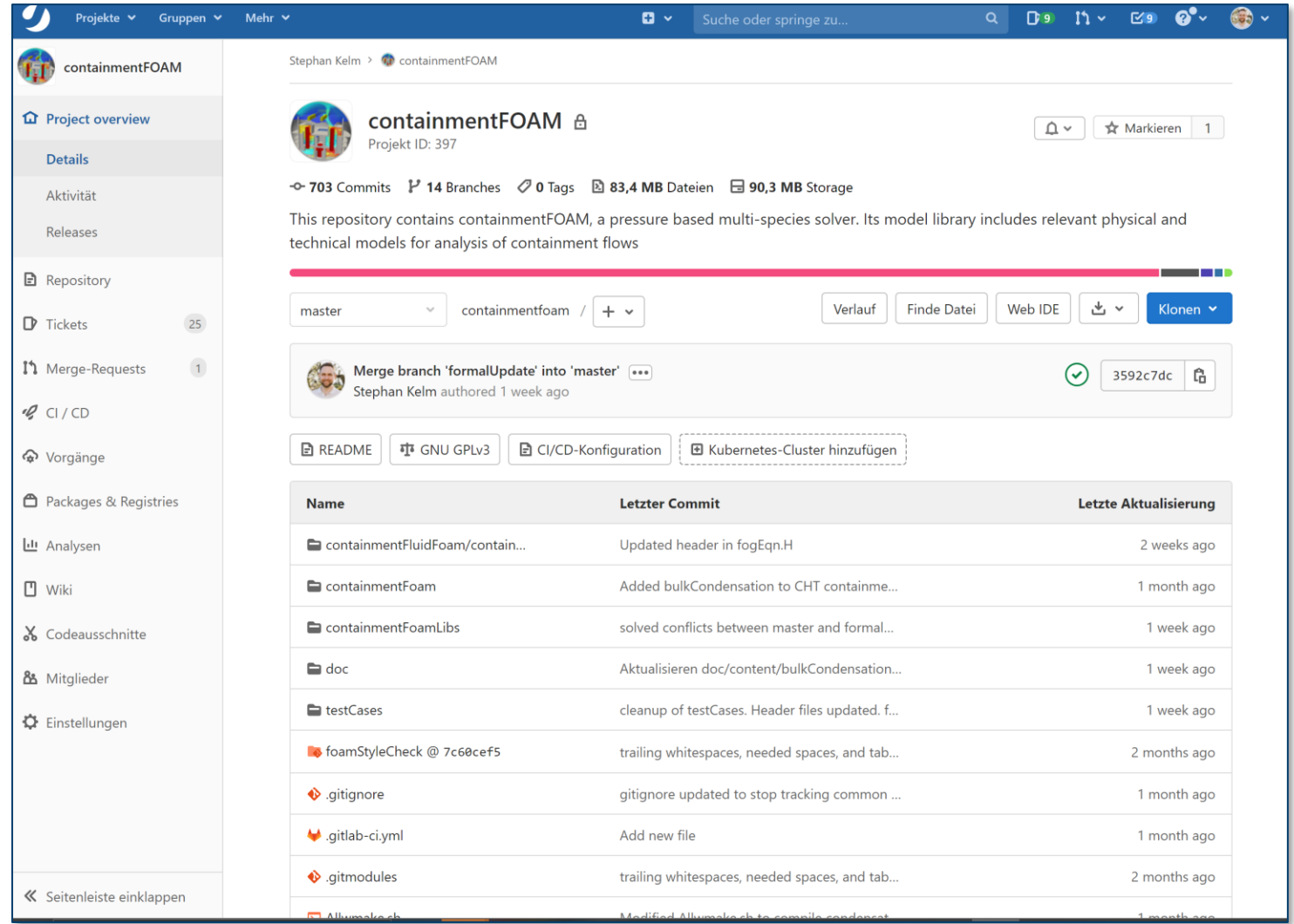
■ Challenges:

- Derivation from / add-on to a famous OS-CFD package enables a broad distribution and usage
 - Known and unknown users → different channels for communication
 - Users with different applications, background and experience (e.g., meshing, CFD, OpenFOAM, containment phenomenology & modeling etc.)
 - Skilled OpenFOAM users may have different established, but ‘incompatible’ workflows, modeling approaches etc.
 - Different applications may exceed the models’ validity range targeted by the developers
 - How to prevent user errors (inconsistent setups, misunderstanding)
- Specific developments are highly complex (product of 4 years PhD projects)
 - How to enable best ‘usability’ of new developments to provide user feedback to the developers ?
 - How to transfer the developers experience to the user?
- Provide and maintain a useful documentation and user guidance
 - Successful usage is important to keep the user's group active
 - Limit time for support

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

-  GitLab platform at FZJ
 - Version management
 - CI/CD Environment
 - Ticket system
 - Wiki (only internal)
 - flexible account management via ‚github accounts‘




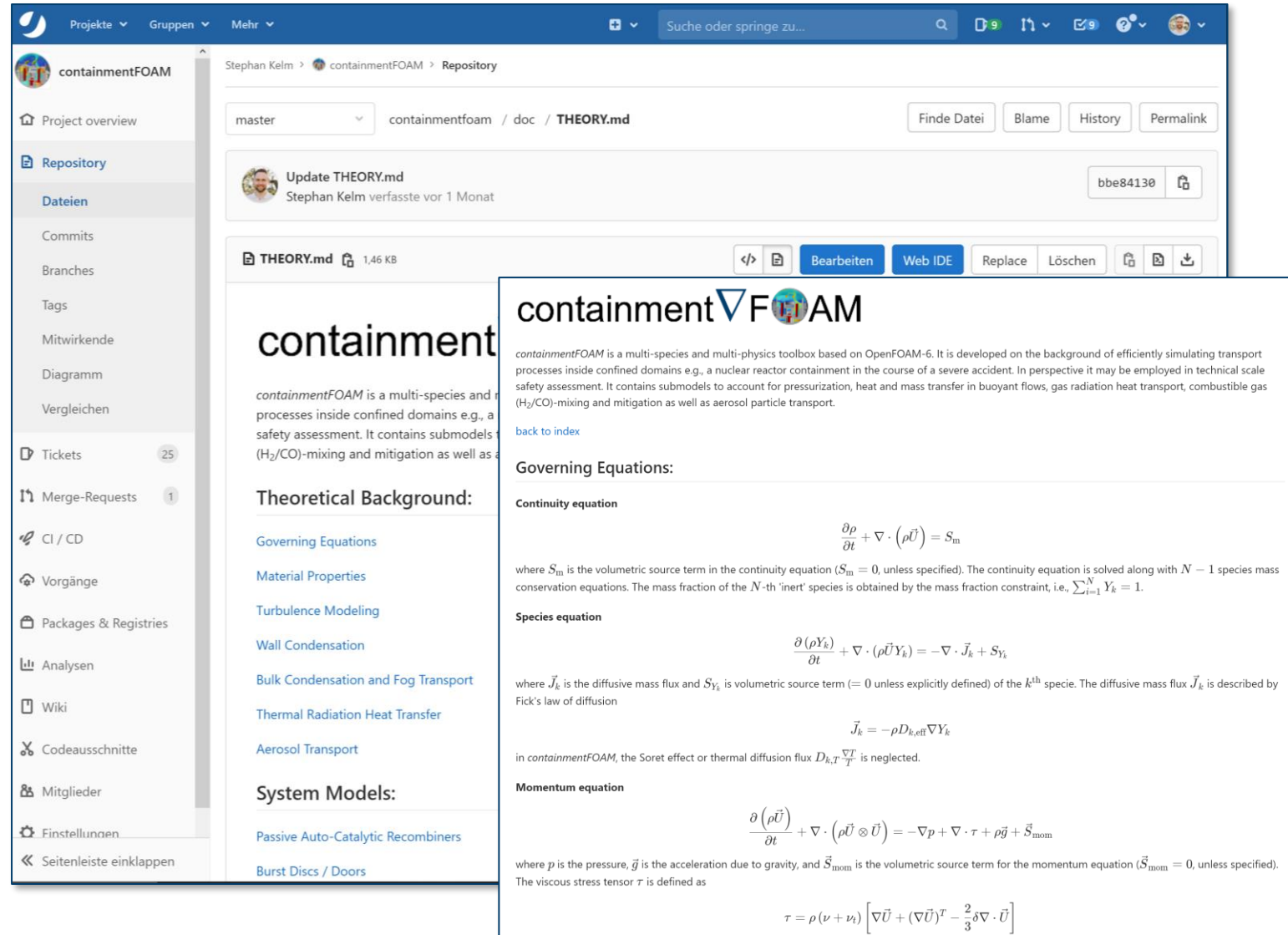
The screenshot displays the GitLab web interface for the 'containmentFOAM' project. The left sidebar shows the project overview, details, repository, tickets (25), merge requests (1), CI/CD, Vorgänge, Packages & Registries, Analysen, Wiki, Codeausschnitte, Mitglieder, and Einstellungen. The main content area shows the project details, including the repository name, project ID (397), and statistics (703 Commits, 14 Branches, 0 Tags, 83,4 MB Dateien, 90,3 MB Storage). A description states: 'This repository contains containmentFOAM, a pressure based multi-species solver. Its model library includes relevant physical and technical models for analysis of containment flows'. Below this, there are buttons for 'Verlauf', 'Finde Datei', 'Web IDE', and 'Klonen'. A merge request is shown: 'Merge branch 'formalUpdate' into 'master'' by Stephan Kelm, authored 1 week ago, with commit ID 3592c7dc. Below the merge request, there are buttons for 'README', 'GNU GPLv3', 'CI/CD-Konfiguration', and 'Kubernetes-Cluster hinzufügen'. A table lists the repository contents with columns for Name, Letzter Commit, and Letzte Aktualisierung.

Name	Letzter Commit	Letzte Aktualisierung
containmentFluidFoam/contain...	Updated header in fogEqn.H	2 weeks ago
containmentFoam	Added bulkCondensation to CHT containme...	1 month ago
containmentFoamLibs	solved conflicts between master and formal...	1 week ago
doc	Aktualisieren doc/content/bulkCondensation...	1 week ago
testCases	cleanup of testCases. Header files updated. f...	1 week ago
foamStyleCheck @ 7c60cef5	trailing whitespaces, needed spaces, and tab...	2 months ago
.gitignore	gitignore updated to stop tracking common ...	1 month ago
.gitlab-ci.yml	Add new file	1 month ago
.gitmodules	trailing whitespaces, needed spaces, and tab...	2 months ago
Alkumakech	Modified Alkumakech to compile condensat...	1 month ago

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

-  GitLab platform at FZJ
 - Version management
 - CI/CD Environment
 - Ticket system
 - Wiki
 - flexible account management
 - GitLab flavored Markdown files
 - Model details



The screenshot displays the GitLab interface for the 'containmentFOAM' repository. The left sidebar contains navigation links for project overview, repository files, commits, branches, tags, contributors, diagrams, and tickets. The main content area shows the 'THEORY.md' file, which is a multi-species and multi-physics toolbox based on OpenFOAM-6. The document content includes a description of the toolbox, a list of governing equations (Continuity, Species, and Momentum), and a list of system models (Passive Auto-Catalytic Recombiners and Burst Discs / Doors).

containmentFOAM

containmentFOAM is a multi-species and multi-physics toolbox based on OpenFOAM-6. It is developed on the background of efficiently simulating transport processes inside confined domains e.g., a nuclear reactor containment in the course of a severe accident. In perspective it may be employed in technical scale safety assessment. It contains submodels to account for pressurization, heat and mass transfer in buoyant flows, gas radiation heat transport, combustible gas (H₂/CO)-mixing and mitigation as well as aerosol particle transport.

[back to index](#)

Governing Equations:

Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = S_m$$

where S_m is the volumetric source term in the continuity equation ($S_m = 0$, unless specified). The continuity equation is solved along with $N - 1$ species mass conservation equations. The mass fraction of the N -th 'inert' species is obtained by the mass fraction constraint, i.e., $\sum_{i=1}^N Y_i = 1$.

Species equation

$$\frac{\partial (\rho Y_k)}{\partial t} + \nabla \cdot (\rho \vec{U} Y_k) = -\nabla \cdot \vec{J}_k + S_{Y_k}$$

where \vec{J}_k is the diffusive mass flux and S_{Y_k} is volumetric source term ($= 0$ unless explicitly defined) of the k -th specie. The diffusive mass flux \vec{J}_k is described by Fick's law of diffusion

$$\vec{J}_k = -\rho D_{k,\text{eff}} \nabla Y_k$$

in *containmentFOAM*, the Soret effect or thermal diffusion flux $D_{k,T} \frac{\nabla T}{T}$ is neglected.

Momentum equation


$$\frac{\partial (\rho \vec{U})}{\partial t} + \nabla \cdot (\rho \vec{U} \otimes \vec{U}) = -\nabla p + \nabla \cdot \tau + \rho \vec{g} + \vec{S}_{\text{mom}}$$

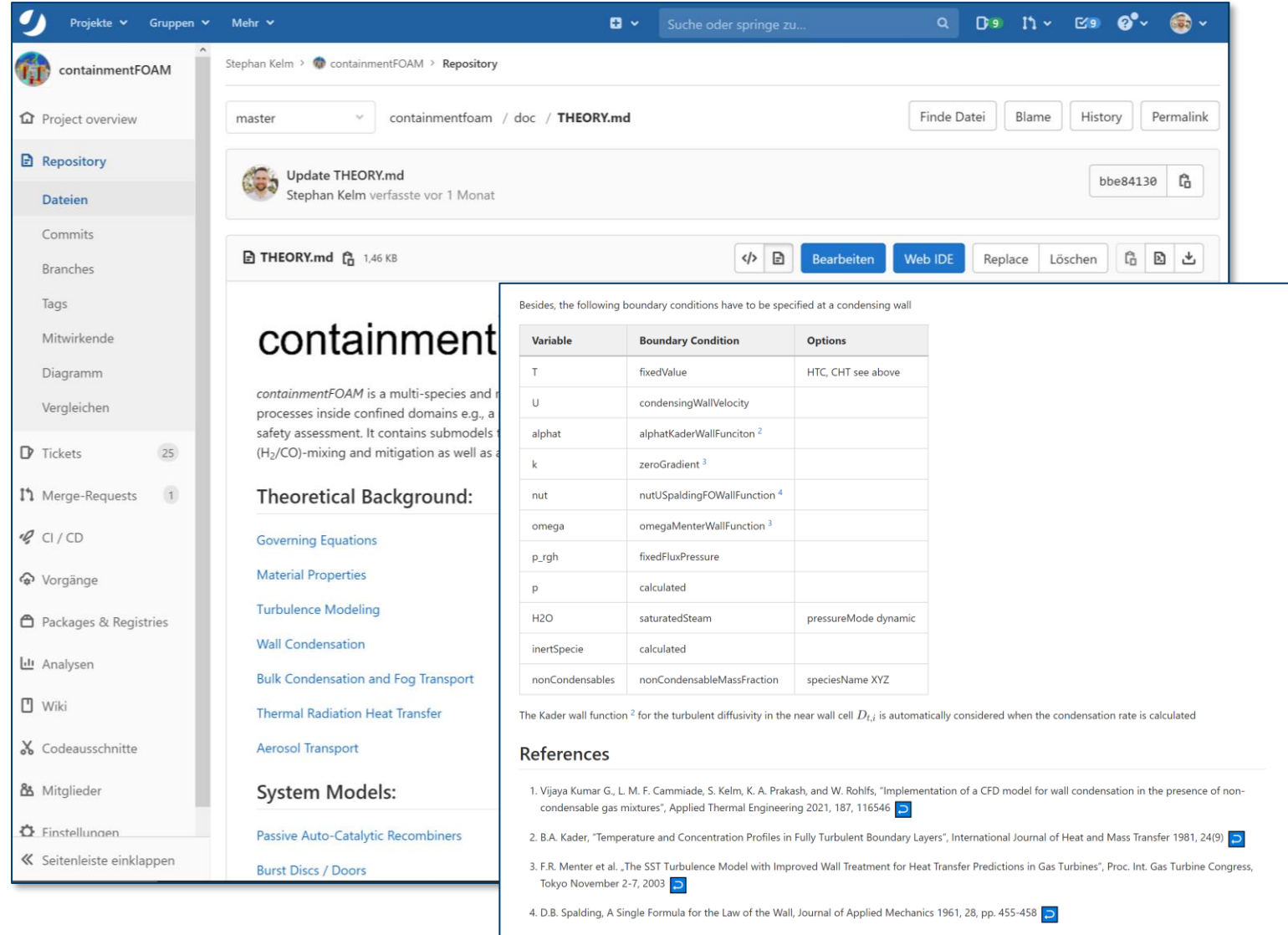
where p is the pressure, \vec{g} is the acceleration due to gravity, and \vec{S}_{mom} is the volumetric source term for the momentum equation ($\vec{S}_{\text{mom}} = 0$, unless specified). The viscous stress tensor τ is defined as

$$\tau = \rho (\nu + \nu_t) \left[\nabla \vec{U} + (\nabla \vec{U})^T - \frac{2}{3} \delta \nabla \cdot \vec{U} \right]$$

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

-  GitLab platform at FZJ
 - Version management
 - CI/CD Environment
 - Ticket system
 - Wiki
 - flexible account management
 - GitLab flavored Markdown files
 - Model details
 - Recommended usage
 - Link to source files
 - References



Stephan Kelm > containmentFOAM > Repository

master containmentfoam / doc / THEORY.md

Update THEORY.md
Stephan Kelm verfasste vor 1 Monat

bbe84130

THEORY.md 1,46 KB

Bearbeiten Web IDE Replace Löschen

containment

containmentFOAM is a multi-species and processes inside confined domains e.g., a safety assessment. It contains submodels (H₂/CO)-mixing and mitigation as well as

Theoretical Background:

- Governing Equations
- Material Properties
- Turbulence Modeling
- Wall Condensation
- Bulk Condensation and Fog Transport
- Thermal Radiation Heat Transfer
- Aerosol Transport

System Models:

- Passive Auto-Catalytic Recombiners
- Burst Discs / Doors

Besides, the following boundary conditions have to be specified at a condensing wall

Variable	Boundary Condition	Options
T	fixedValue	HTC, CHT see above
U	condensingWallVelocity	
alphat	alphatKaderWallFunction ²	
k	zeroGradient ³	
nut	nutUSpaldingFOWallFunction ⁴	
omega	omegaMenterWallFunction ³	
p_rgh	fixedFluxPressure	
p	calculated	
H2O	saturatedSteam	pressureMode dynamic
inertSpecie	calculated	
nonCondensables	nonCondensableMassFraction	speciesName XYZ

The Kader wall function² for the turbulent diffusivity in the near wall cell $D_{t,i}$ is automatically considered when the condensation rate is calculated


References

1. Vijaya Kumar G., L. M. F. Camiade, S. Kelm, K. A. Prakash, and W. Rohlf, "Implementation of a CFD model for wall condensation in the presence of non-condensable gas mixtures", Applied Thermal Engineering 2021, 187, 116546 [🔗](#)
2. B.A. Kader, "Temperature and Concentration Profiles in Fully Turbulent Boundary Layers", International Journal of Heat and Mass Transfer 1981, 24(9) [🔗](#)
3. F.R. Menter et al., "The SST Turbulence Model with Improved Wall Treatment for Heat Transfer Predictions in Gas Turbines", Proc. Int. Gas Turbine Congress, Tokyo November 2-7, 2003 [🔗](#)
4. D.B. Spalding, A Single Formula for the Law of the Wall, Journal of Applied Mechanics 1961, 28, pp. 455-458 [🔗](#)

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

■ Communication channels:

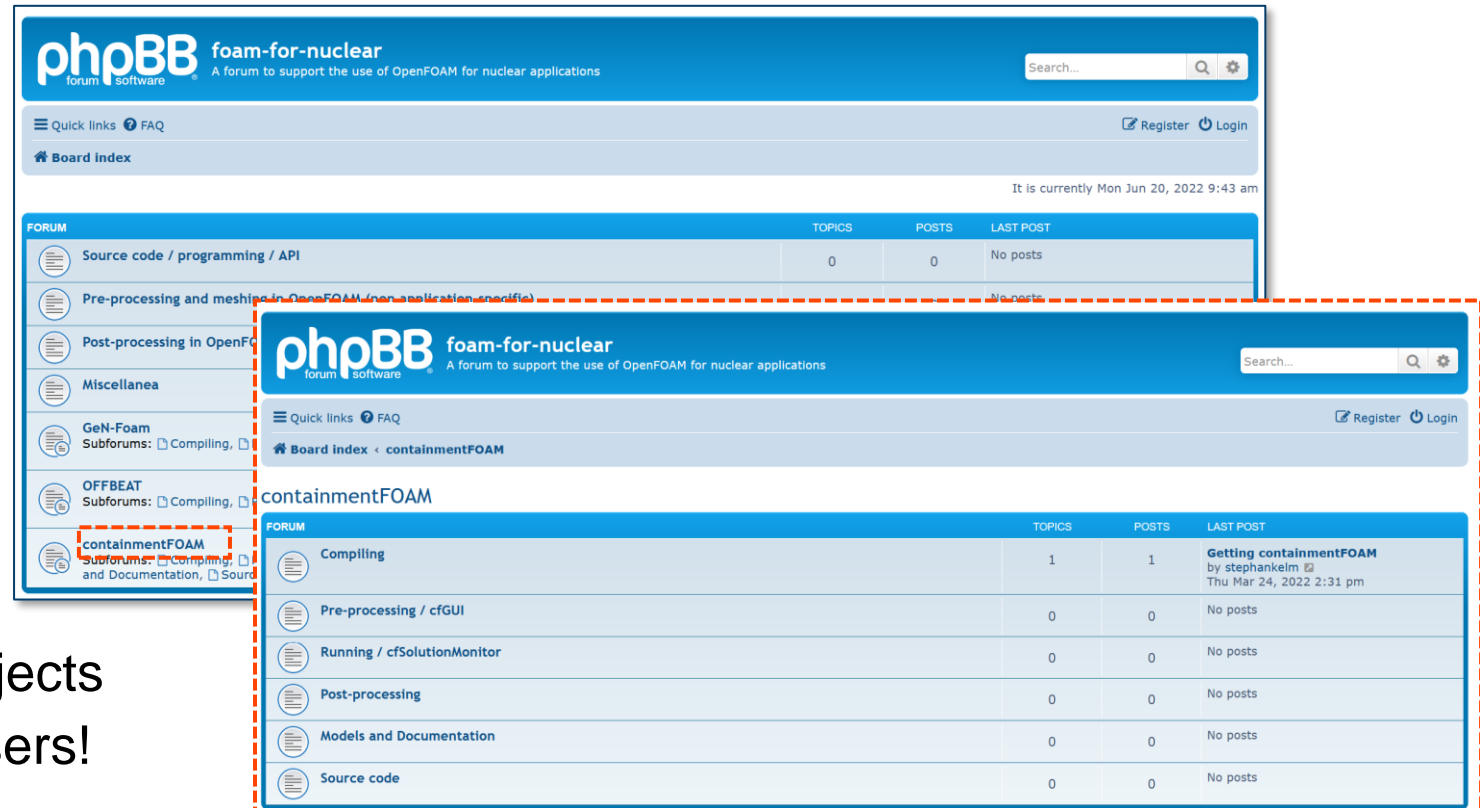
-  chat system (upon request / invitation)
Mattermost®

- Discussion forum


- <https://foam-for-nuclear.org/phpBB/>
thanks to Carlo Fiorina for hosting it !

-  containmentfoam@fz-juelich.de

➤ User support is mandatory, as the projects success depends on the number of users!



The right screenshot displays the 'containmentFOAM' subforum with the following table:

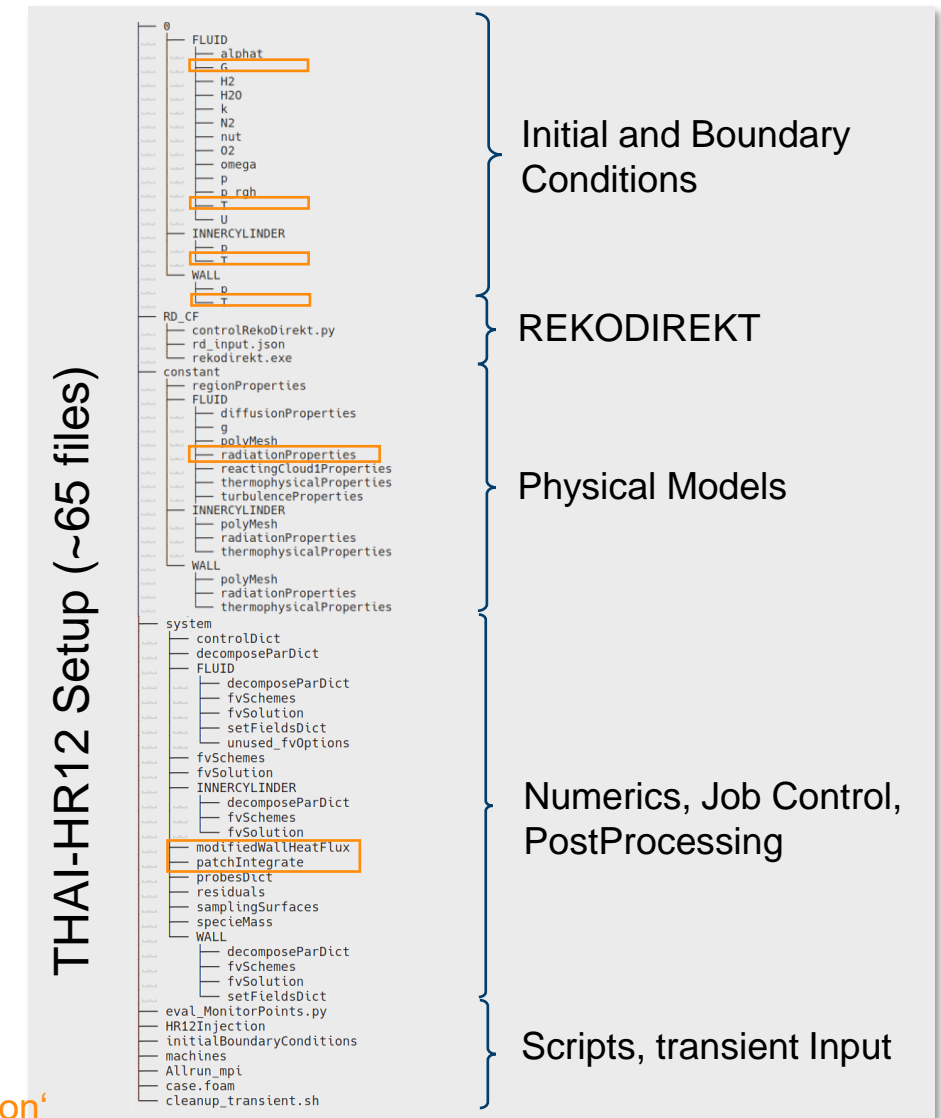
FORUM	TOPICS	POSTS	LAST POST
Compiling	1	1	Getting containmentFOAM by stephankelm  Thu Mar 24, 2022 2:31 pm
Pre-processing / cfGUI	0	0	No posts
Running / cfSolutionMonitor	0	0	No posts
Post-processing	0	0	No posts
Models and Documentation	0	0	No posts
Source code	0	0	No posts

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

- Containment analysis comprises interaction of multiple physical phenomena and safety systems:
 - Definition in separate files (,dictionary‘)
 - Dependencies among dictionaries (and models)
 - Inconsistent definitions possible, which may not cause a crash
 - Broad variety of numerical methods and schemes
- ,Standard‘ & Best Practice required:
 - Prevent input errors
 - Ensure consistent model application
 - Enable comparable and reproducible analysis
 - Support / bug identification and fixing.

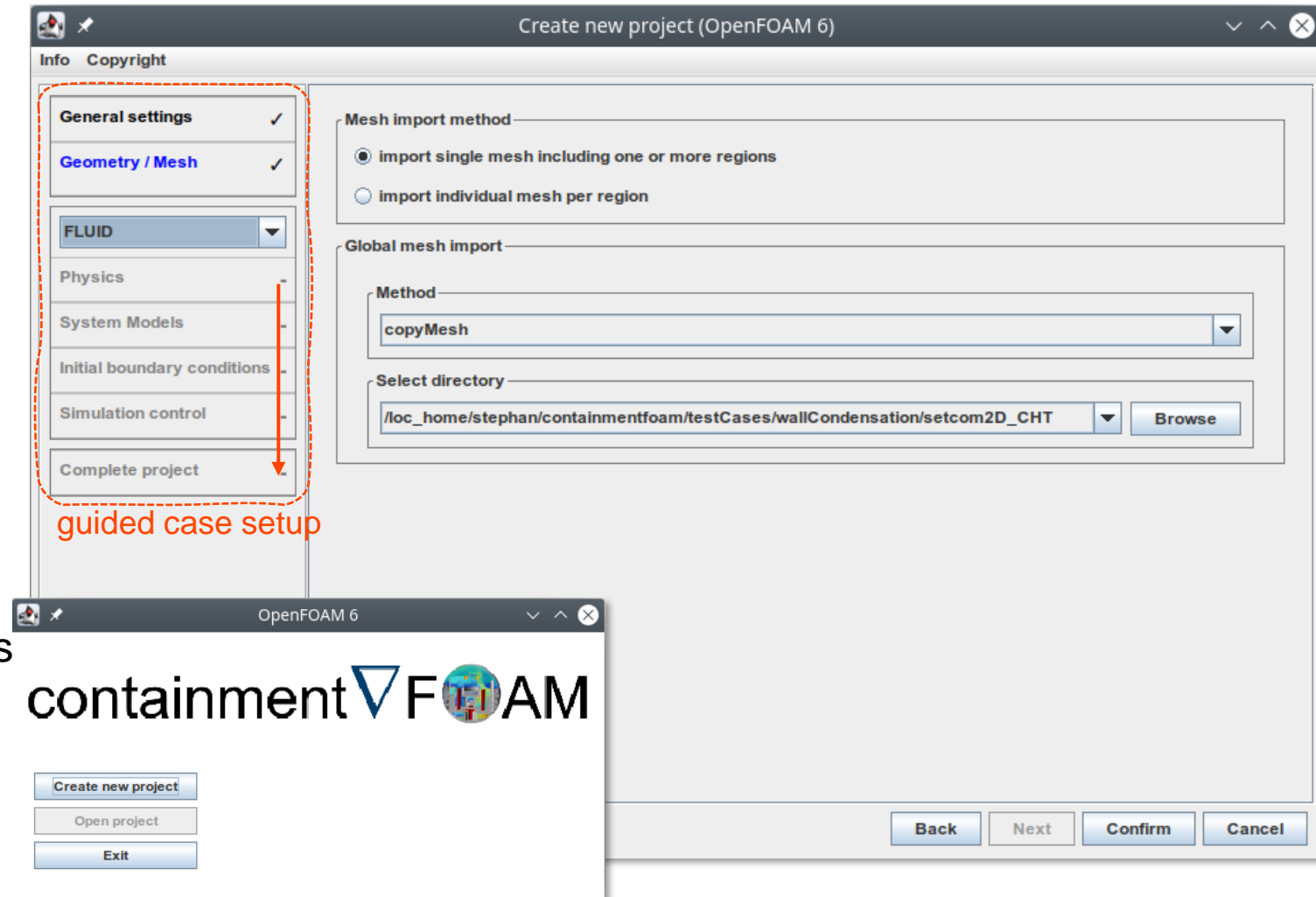
Definition ,gas radiation‘



UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

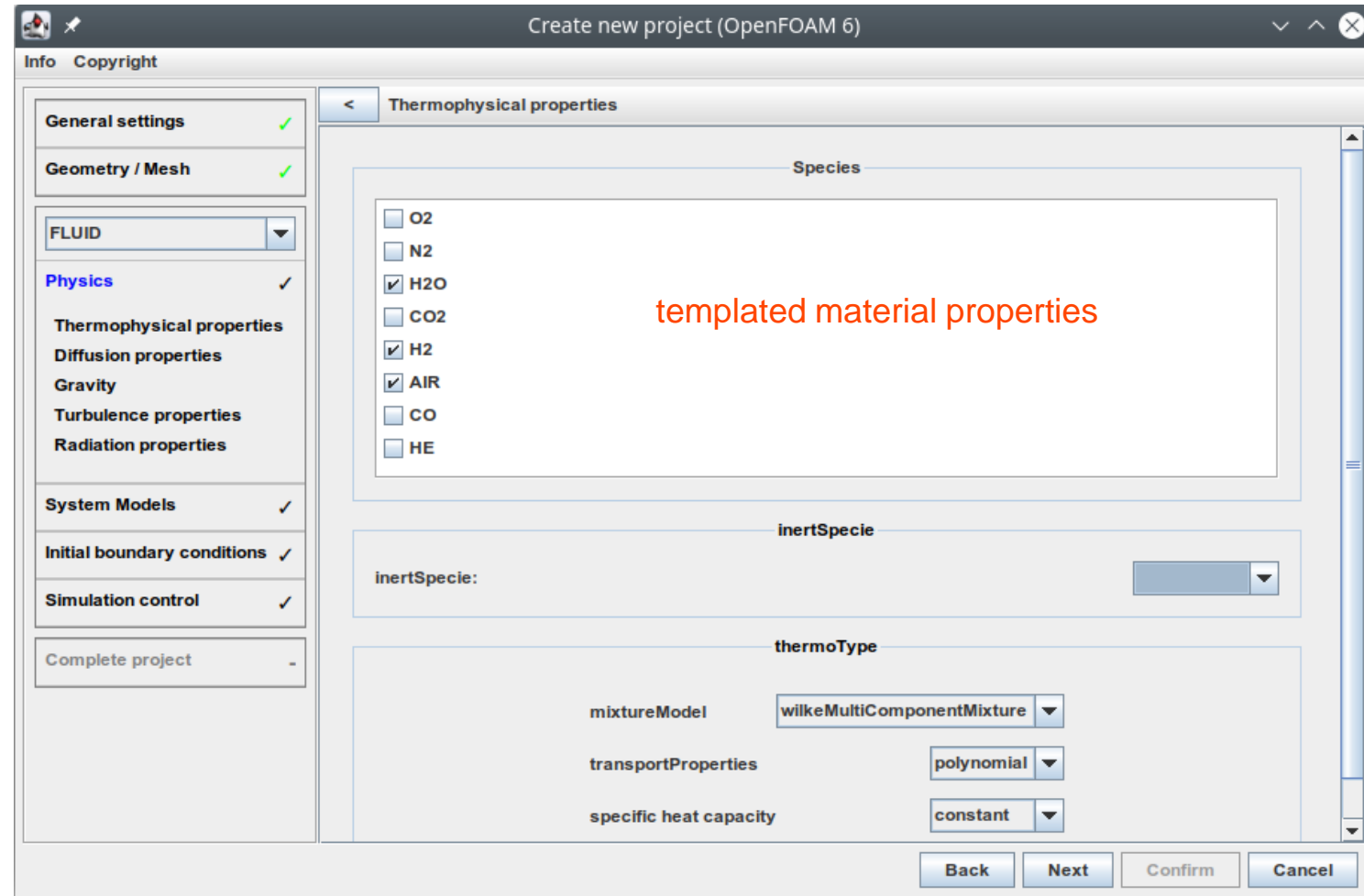
- General idea:
 - Workflow, structure and syntax close to OpenFOAM
 - limit functionality to baseline model and fundamental functions
 - Enable easy entry for new users (only a mesh is needed)
 - Enable fallback for bug tracking
- Mesh import
 - Using OpenFOAM® utilities and libraries
 - Reads mesh quality metrics for further use (e.g. numerics settings)



UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

- Mesh import
- Templated material properties
 - Calculator
 - Polynomial fits (NIST data)



Create new project (OpenFOAM 6)

Info Copyright

General settings ✓

Geometry / Mesh ✓

FLUID

Physics ✓

Thermophysical properties

Diffusion properties

Gravity

Turbulence properties

Radiation properties

System Models ✓

Initial boundary conditions ✓

Simulation control ✓

Complete project -

Thermophysical properties

Species

☐ O2

☐ N2

☒ H2O

☐ CO2

☒ H2

☒ AIR

☐ CO

☐ HE

templated material properties

InertSpecie

inertSpecie:

thermoType

mixtureModel wilkeMultiComponentMixture

transportProperties polynomial

specific heat capacity constant

Back Next Confirm Cancel

UNIQUE OPEN SOURCE ISSUES

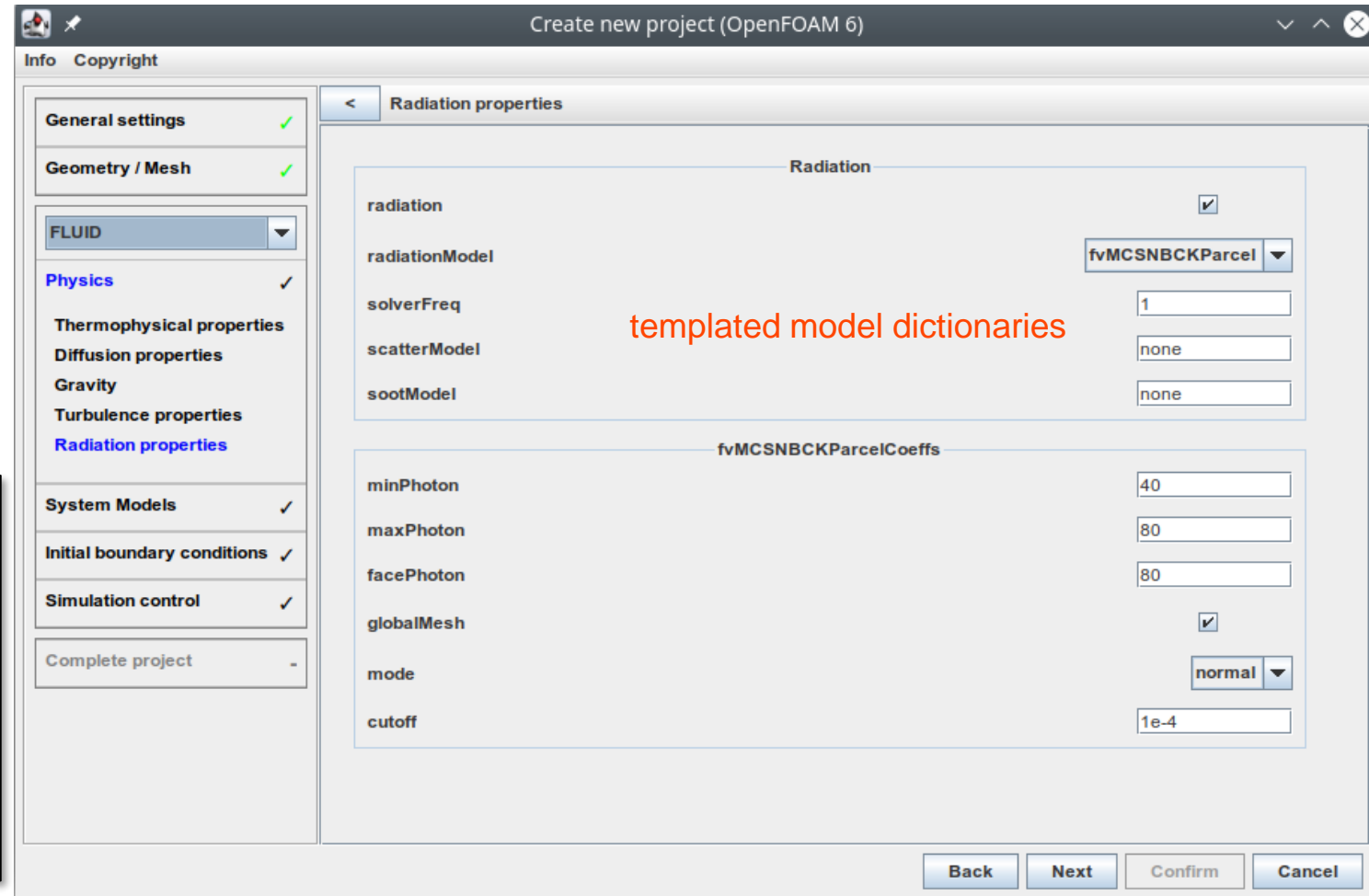
Developer - User Interaction

- Mesh import
- Templated material properties
- Flexible model templates based on JSON, logical rules and tooltips by the containmentFOAM developers (no Java skills needed)

```
1 "fvMCMParcelCoeffs":{
2   "__settings__":["type=oneLineValue"],
3
4   "__values__":
5   [
6     ["_show_{Label(text=minPhoton)}","_show_{TextField(text=4)}"],
7     ["_show_{Label(text=maxPhoton)}","_show_{TextField(text=50)}"],
8     ["_show_{Label(text=facePhoton)}","_show_{TextField(text=1)}"],
9     ["_show_{Label(text=cutoff)}","_show_{TextField(text=1e-2)}"]
10  ],
11 },
12
13 "absorptionEmissionModel":{
14   "__settings__":["type=oneLineValue"],
15
16   "__values__":
17   [
18     ["_show_{Label(text=absorptionEmissionModel)}","_show_{ChoiceBox(text=[constantAbsorptionEmission,nongreyMeanAbsorptionEmission])}"],
19     ["_show_{Label(text=facePhoton)}","_show_{TextField(text=1)}"],
20     ["_show_{Label(text=cutoff)}","_show_{TextField(text=1e-2)}"]
21  ],
22 }
```

default parameters

model selection

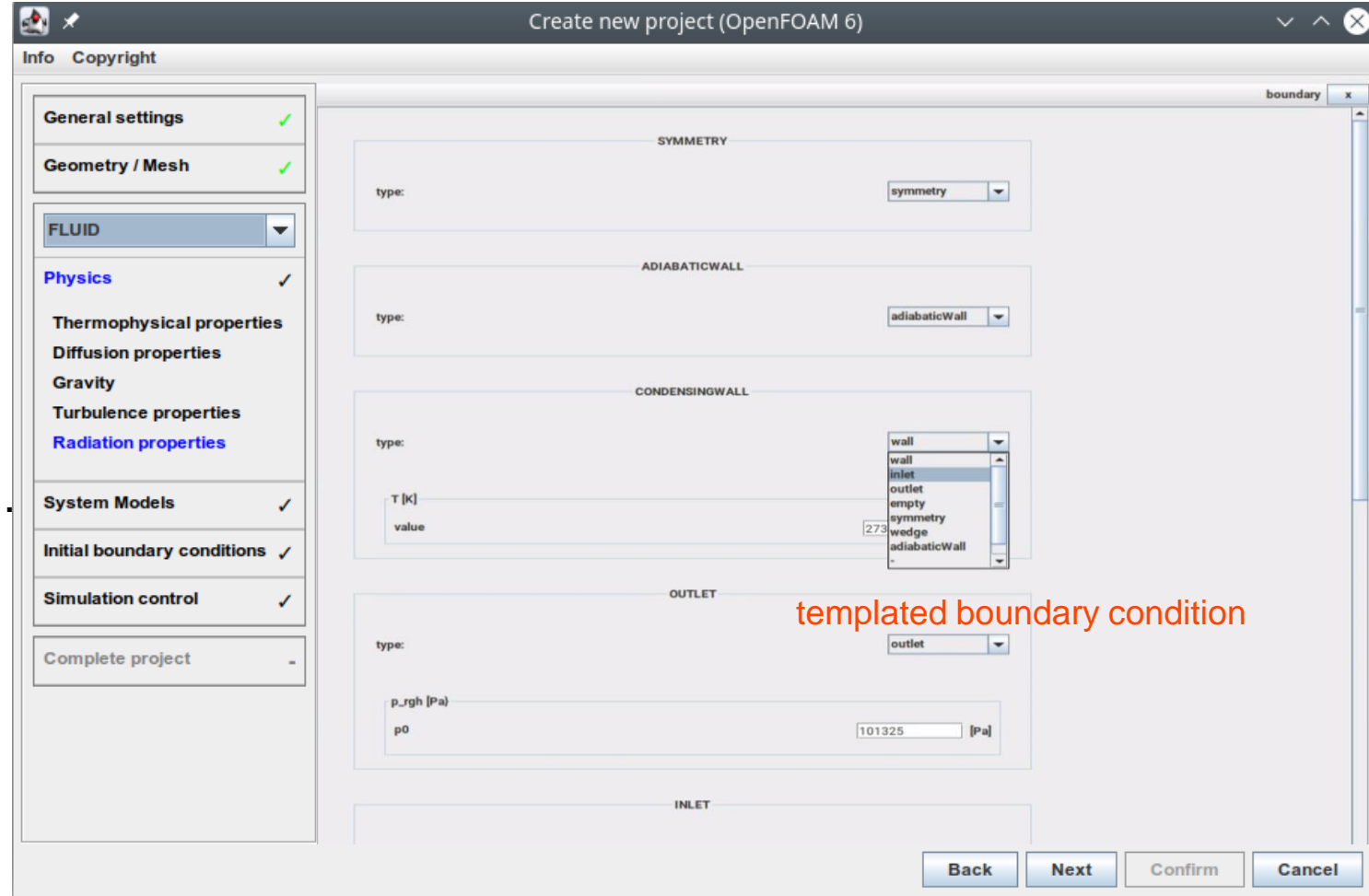


templated model dictionaries

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

- Mesh import
- Templated material properties
- Flexible extension
- Consistent IC & BC specification
 - Predefined boundary types
(,condensing wall', ,inlet' etc.)
 - csv import, table editor, global variables..

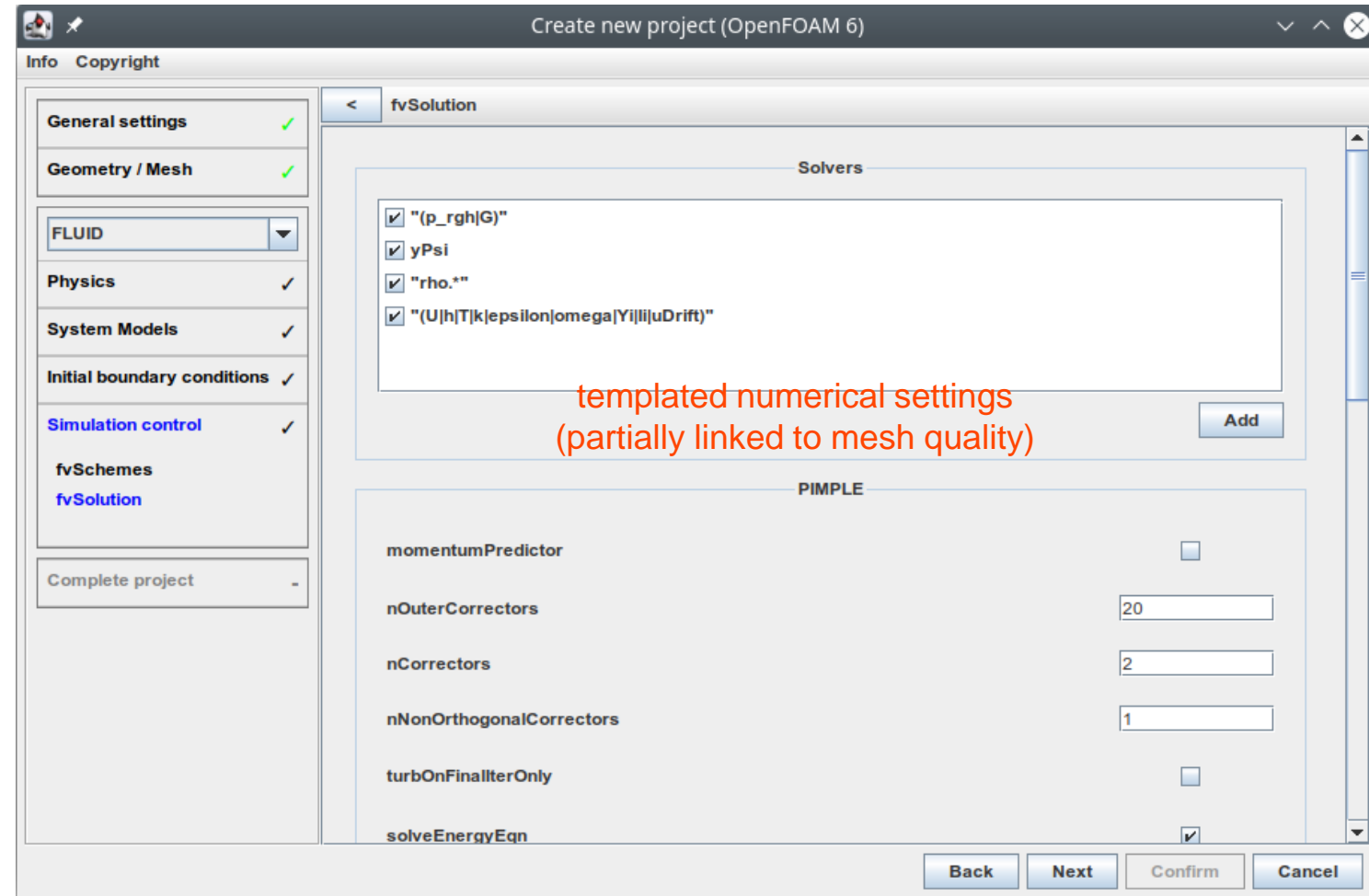


templated boundary condition

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

- Mesh import
- Templated material properties
- Flexible extension
- Consistent IC & BC specification
- Numerics and simulation control
 - Best practice for numerical settings (fvSchemes and fvSolution)



Create new project (OpenFOAM 6)

Info Copyright

General settings ✓

Geometry / Mesh ✓

FLUID

Physics ✓

System Models ✓

Initial boundary conditions ✓

Simulation control ✓

fvSchemes

fvSolution

Complete project -

fvSolution

Solvers

- ☒ "(p_rgh|G)"
- ☒ yPsi
- ☒ "rho.*"
- ☒ "(U|h|T|k|epsilon|omega|Yi|li|uDrift)"

templated numerical settings
(partially linked to mesh quality)

Add

PIMPLE

momentumPredictor ☐

nOuterCorrectors 20

nCorrectors 2

nNonOrthogonalCorrectors 1

turbOnFinalIterOnly ☐

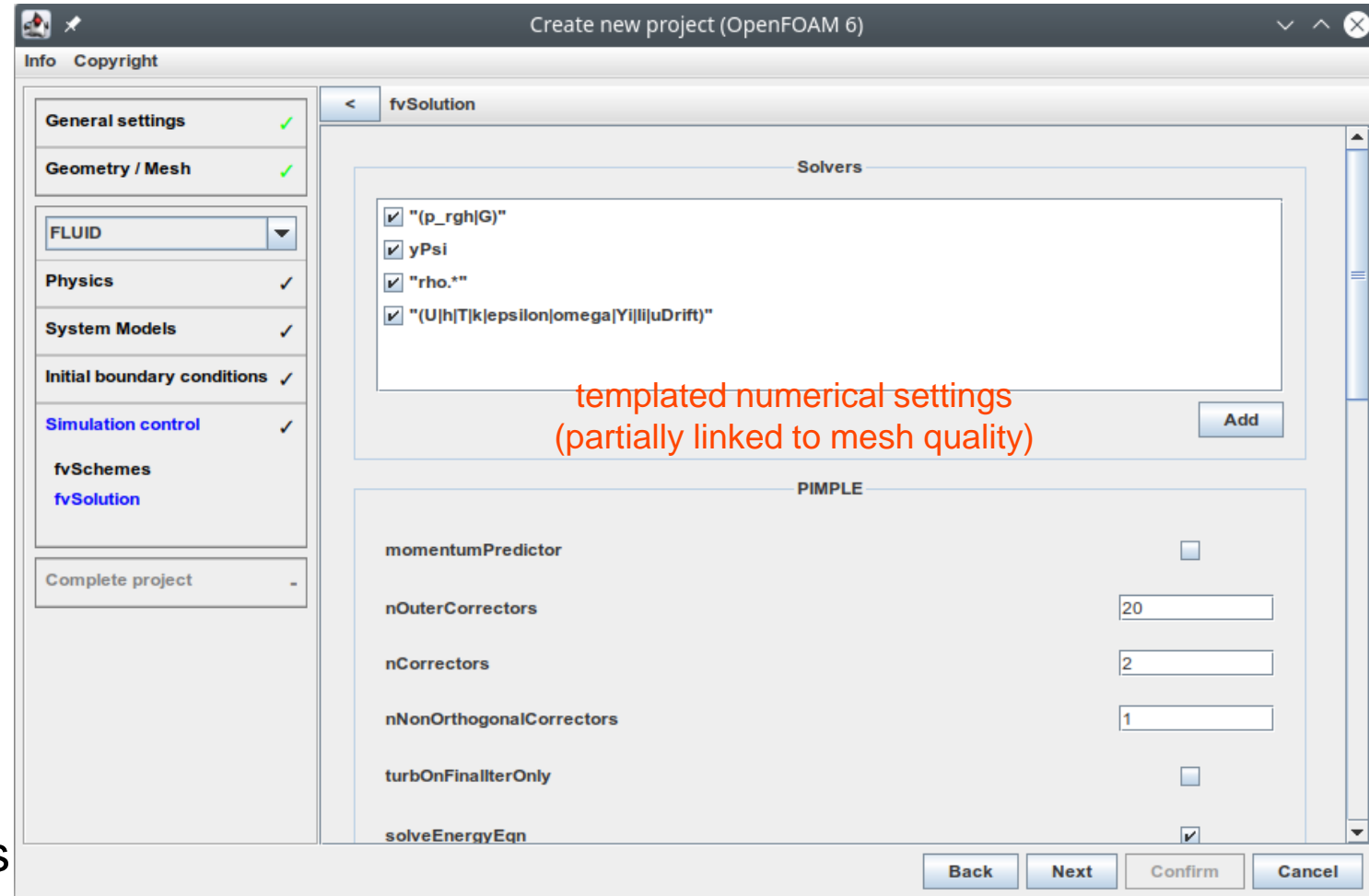
solveEnergyEqn ☒

Back Next Confirm Cancel

UNIQUE OPEN SOURCE ISSUES

Developer - User Interaction

- Mesh import
- Templated material properties
- Flexible extension
- Consistent IC & BC specification
- Numerics and simulation control
 - Best practice for numerical settings (fvSchemes and fvSolution)
 - Include predefined functionObjects (e.g. mass/energy Balance)
- Fast, reproducible and verified case setup is fundamental requirement for code use, support and trustful analysis



Create new project (OpenFOAM 6)

Info Copyright

General settings ✓

Geometry / Mesh ✓

FLUID

Physics ✓

System Models ✓

Initial boundary conditions ✓

Simulation control ✓

fvSchemes

fvSolution

Complete project -

fvSolution

Solvers

- ☒ "(p_rgh|G)"
- ☒ yPsi
- ☒ "rho.*"
- ☒ "(U|h|T|k|epsilon|omega|Yi|li|uDrift)"

templated numerical settings
(partially linked to mesh quality)

Add

PIMPLE

momentumPredictor ☐

nOuterCorrectors 20

nCorrectors 2

nNonOrthogonalCorrectors 1


turbOnFinalIterOnly ☐

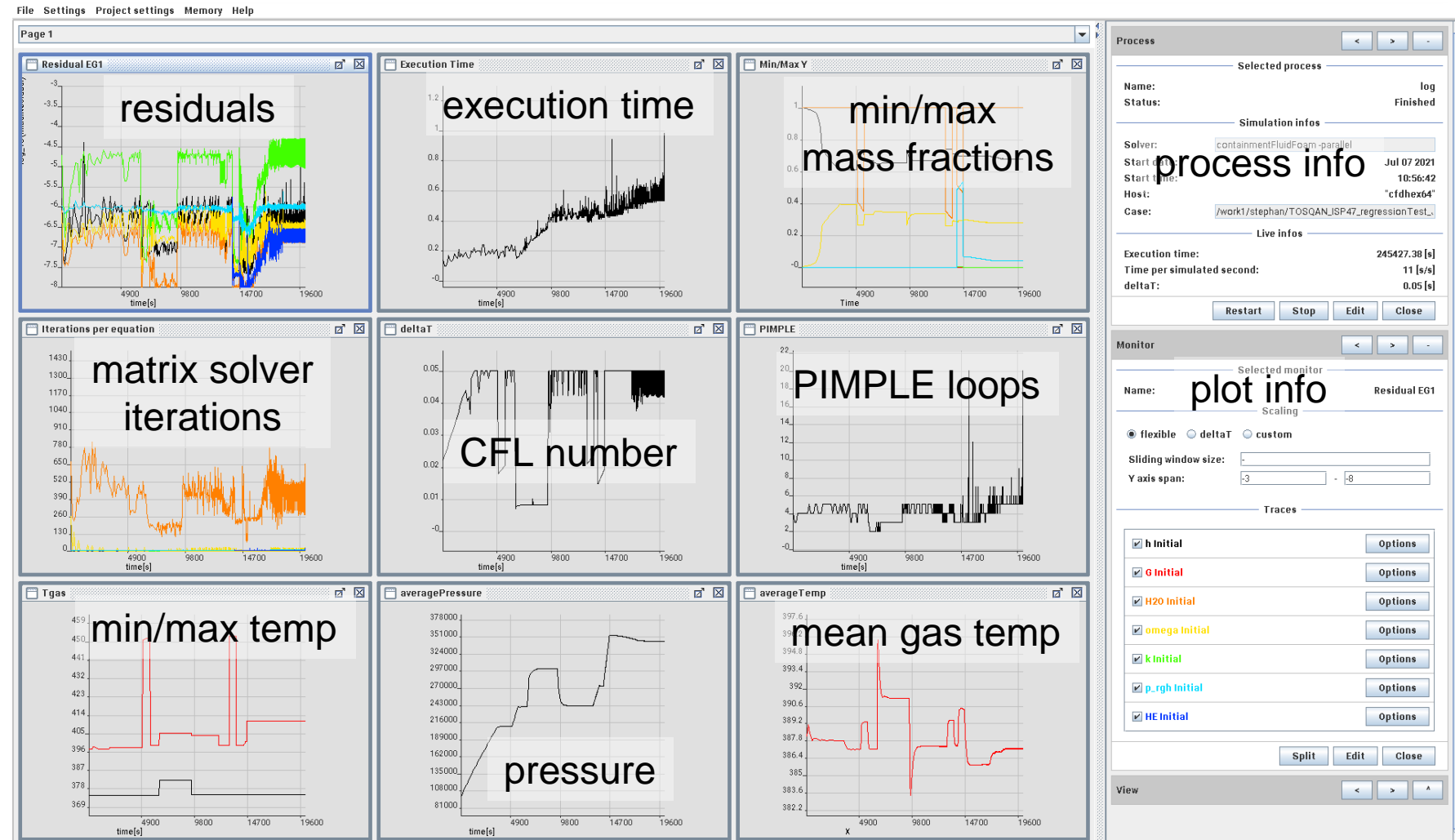
solveEnergyEqn ☒

Back Next Confirm Cancel

UNIQUE OPEN SOURCE ISSUES


Solution Monitor

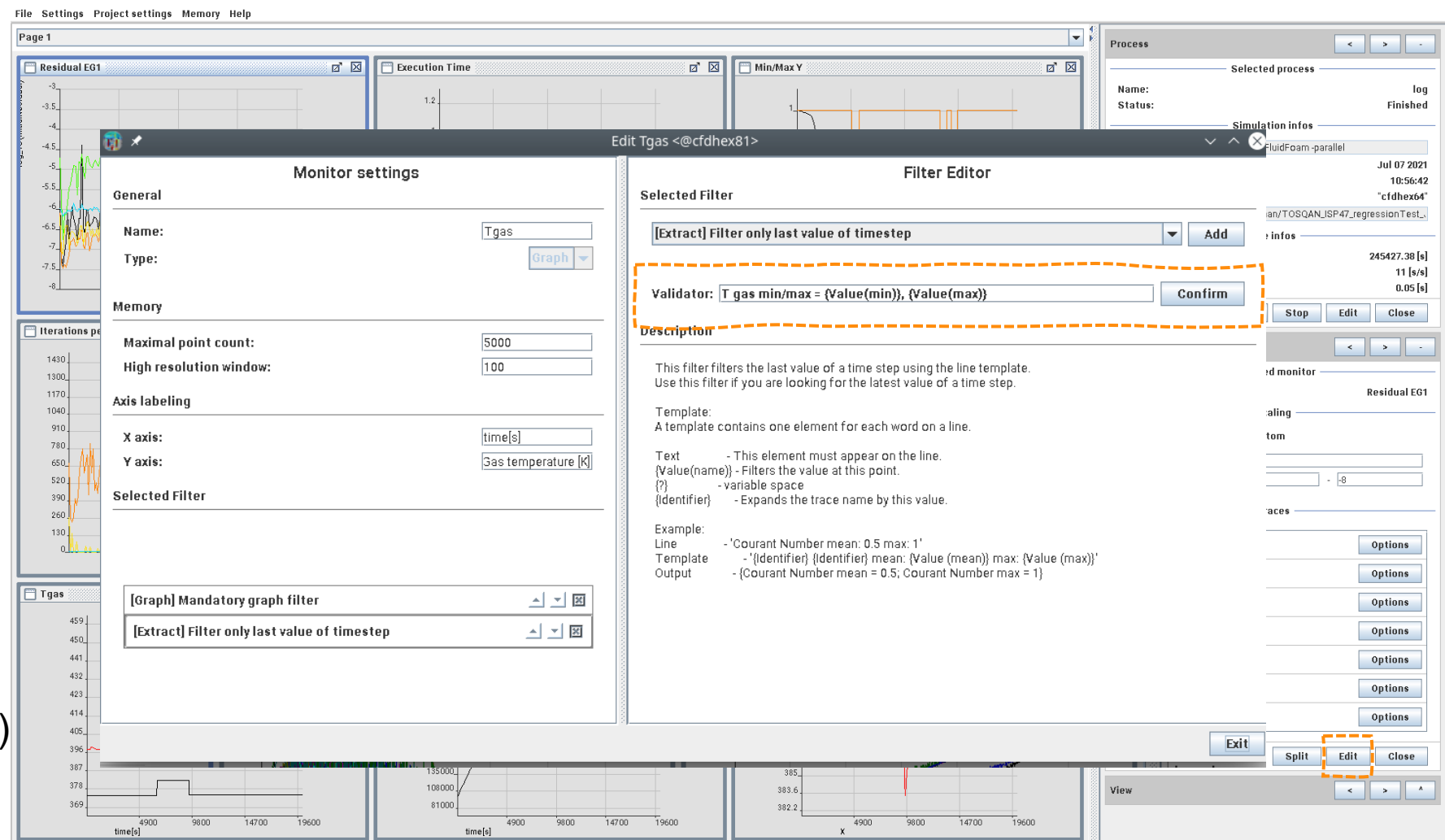
- Good setup an then ?
 - Solution Monitoring!
- Based on **OpenJDK** 
and jchart2D
(<http://jchart2d.sourceforge.net/>)
 - Streams OpenFOAM logs and functionObject output
 - Grid and tabbed view




CONTAINMENTFOAM FRAMEWORK

Solution Monitor

- Based on **OpenJDK**  and **jchart2D**
(<http://jchart2d.sourceforge.net/>)
- Steams OpenFOAM logs and functionObject output
- Tab and Grid view
- Flexible Regular Expression syntax
- Filters, e.g. floating average, exponential smoothing, FFT..
- Edit setup at runtime (selected dictionaries only)



- Introduction
 - Background and Motivation
 - Strategy and General Considerations
 - Status of containment  VFAAM
- Open-source challenges
 - Maintenance, distribution
 - User-Developer interaction
- Best practices
- Summary and Perspective

Our experiences

- Documentation:
 - Do it in parallel to code development, e.g. using gitlab flavored markdown.
- Requirements:
 - Export control & dual use: needs time and careful argumentation with non-technical persons
 - Think of (internal and external) users!
 - Think of maintainability: aim for code-quality, harmonized code style, best practices etc.

BEST PRACTICES IN NUCLEAR OS-PROJECTS

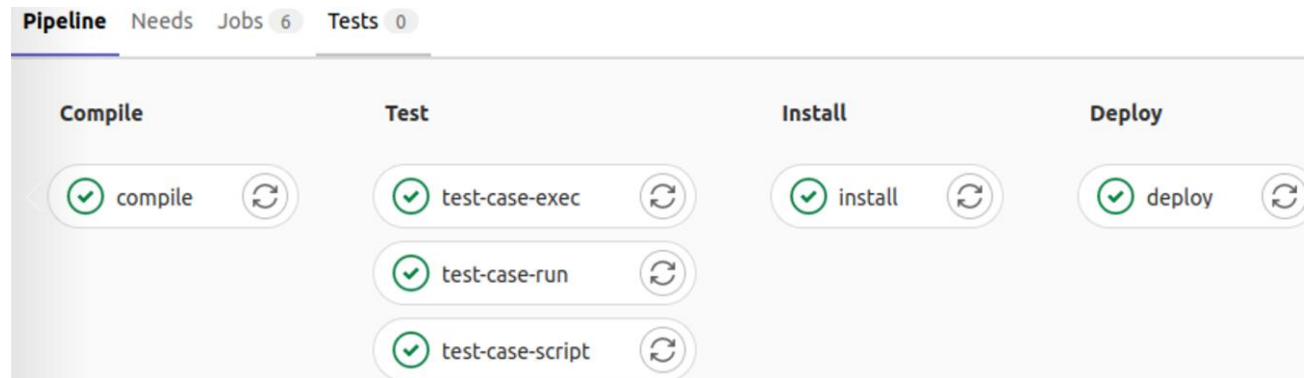
Our experiences


■ Management:

- Currently, in personal union with group lead
- Future, group of (internal and external) contributors


■ Testing / CI:

- Build test + simple test cases via CI pipeline; complex (costly) cases executed manually
- Merge from 'feature' to 'development branch' only after independent review and testing (git 'approval')



- Introduction
 - Background and Motivation
 - Strategy and General Considerations
 - Status of containment  VFAAM
- Open-source challenges
 - Maintenance, distribution
 - User-Developer interaction
- Best practices
- Summary and Perspective

SUMMARY AND CONCLUSIONS

- Development of containment  as an OpenFOAM add-on for containment flows since 2017
- Coordinated effort to enable integration and balance of individual sub models and maintenance under the different time horizon of the individual contributors
- Productive version was released in 2021 – <https://go.fzj.de/containmentFOAM>, currently small & known users group
- Measures taken to enable a productive user-experience and promoting a larger users group
 - Parallel development of Java based tools to assist users:
 - cfGUI – templated workflow to create new simulation setups
 - cfSolutionMonitor – graphical visualization of simulation logs for run-time verification and analysis
 - Communication via Mattermost chat, discussion forum
- Several best practices for software development employed, but limits exist due to manpower
- Applications: PWR flammable gas control (EU-AMHYCO), iPWR containment cooling (EU-SASPAM)

ACKNOWLEDGEMENTS

- German Federal Ministry for the Environment, Nature Conservation, Nuclear Safety and Consumer Protection for funding code maintenance and integration to the national CFD reference package CF2REF (Fkz **150 1633B**) as well as the ongoing projects UQ4CFD (Fkz **150 1595**) and SETCOM-2 on wall condensation modeling (Fkz **150 1591**)
- Former German Federal Ministry of Economic Affairs and Energy (BMWi) and for funding projects related to further development of CFD for reactor safety application (Fkz **150 1407**) and the REKO experimental and model development program (Fkz **150 1308 / 150 1394 / 150 1470**) as well as the SETCOM projects (Fkz **1501404 / 1501489**)
- Helmholtz Interdisciplinary Doctoral Training in Energy and Climate Research (HITEC) Graduate School, the German Academic Exchange Service (DAAD) and the Chinese Scholarship Council (CSC) for funding PhD positions and student exchange
- Prof. K. Arul Prakash, Indian Institute of Technology Madras, India (IITM) for collaborating under IITM-RWTH Joint Doctoral Programme, DAAD IIT-MSP Programmes.
- Our contributors, partners and colleagues and beta testers
- The original developers, maintainers and contributors of OpenFOAM®

Supported by:



Federal Ministry
for the Environment, Nature Conservation,
Nuclear Safety and Consumer Protection

based on a decision of
the German Bundestag

Supported by:



Federal Ministry
for Economic Affairs
and Climate Action

on the basis of a decision
by the German Bundestag



- BACKUP

■ Design of containmentFOAM

- Not a stand-alone software, but add-on to OpenFOAM
- new functionality is added via
 - cloned and modified code (minor extensions)
 - separate base/derived classes (e.g., condensation)
 - Access information from OpenFOAM
 - Conducting plausibility checks during instantiation
 - Doing the math
 - Provide access functions to solvers/models
 - Encapsulated for better maintainability

```
FatalErrorInFunction
<< " Illegal boundary condition for " << U.name() << " field boundary = "
<< mesh.boundaryMesh()[patchi].name()
<< " . Select 'condensingWallVelocity' boundary condition"
<< " .Wall condensation error "
<< exit(FatalError);
```

plausibility check for model use and defined boundary conditions

```
//- Update fog velocity
condensation->bulkUFogCorrect();

//- Compute sum of uFog and uDrift volumetric flux fields
phiFog = fvc::flux(condensation->bulkUFog()) + fvc::flux(condensation->bulkUDrift(g));

fvScalarMatrix fogEqn
(
    fvm::ddt(fog)
    + fvm::div(phiFog, fog, "div(phi,fog)")
    ==
    fvm::laplacian((massDiffusion->Dt(0) + condensation->bulkDBrownian()), fog)
    + condensation->bulkMassTransferRate()
);
```

access functions used in the solver. Here fogEqn.H