IAEA
International Atomic Energy Agency

Multi-physics modeling and simulation of nuclear reactors using OpenFOAM

30 Aug 2022 – 6 October 2022 (every Tuesday & Thursday)

Contact: ONCORE@iaea.org

Introduction to GeN-Foam - Theory

Carlo Fiorina

About these two lectures

What to expect

- A crash introduction to GeN-Foam: theory and practice

What not to expect

- A full course on the multi-physics analysis of nuclear reactors
- A full course on the use of GeN-Foam

Objectives

- Brief recap of multi-physics modelling of nuclear reactors
- Description of the basics structure of GeN-Foam
- Understanding of modelling capabilities of GeN-Foam and its pros & cons
- How to approach GeN-Foam
- References, keywords, best practices that can simplify an autonomous learning of GeN-Foam

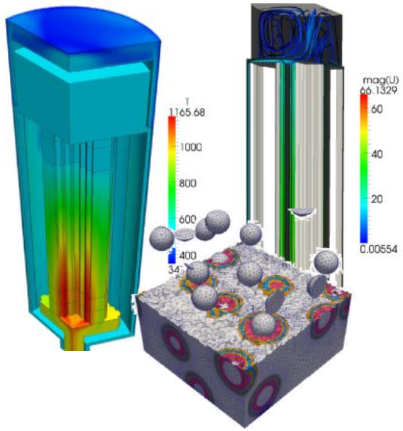
Warning: some slides with a lot of text. This is meant for autonomous use after the lecture.

Use of OpenFOAM for nuclear multi-physics

2000-2010
First activities

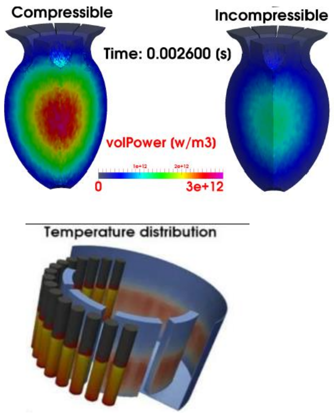
2010-2015
First widespread use

2015-2021
First coordinated and persistent developments

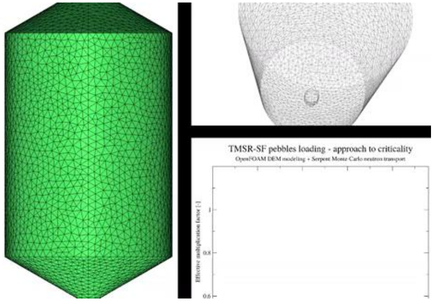


Pebble bed and prismatic HTGRs

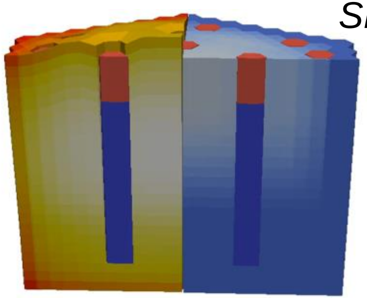
MSRs

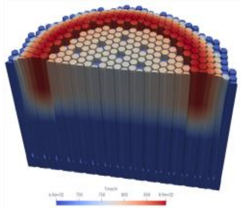


FHRs

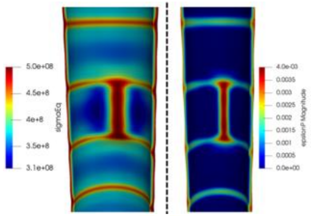


SFRs

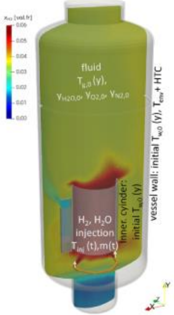




GeN-Foam



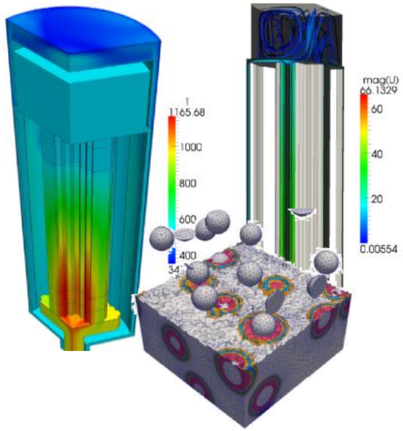
Fuel Behaviour (OFFBEAT)



Containment Flows containmentFoam

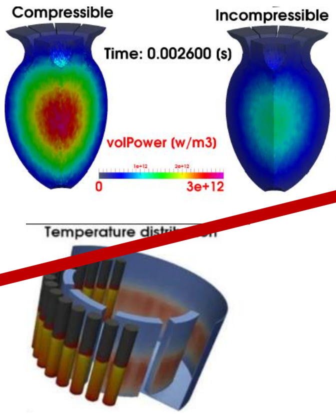
Use of OpenFOAM for nuclear multi-physics

2000-2010	2010-2015	2015-2021
First activities	First widespread use	First coordinated and persistent developments

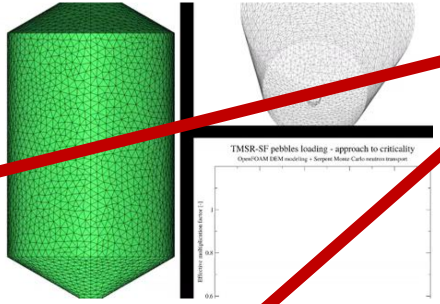


Pebble bed and prismatic HTGRs

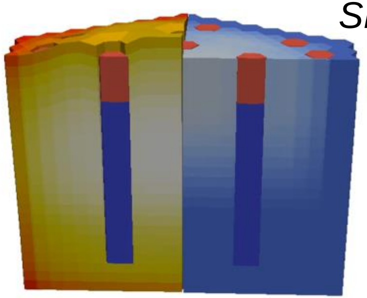
MSRs

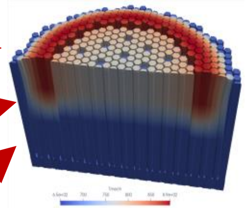


FHRs

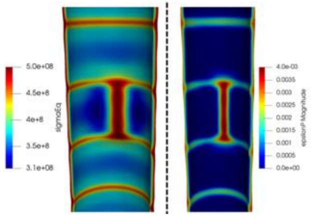


SFRs

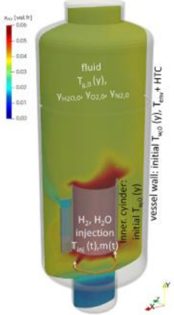




GeN-Foam



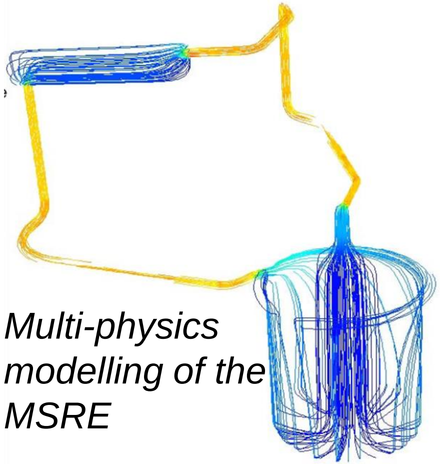
Fuel Behaviour (OFFBEAT)



Containment Flows containmentFoam

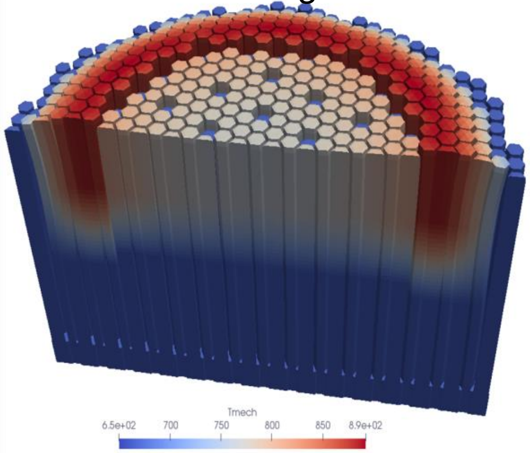
GeN-Foam: Generalized Nuclear Field operation and manipulation

- **Since 2014, EPFL + PSI + contributions from various institutions**
- **Developed to complement legacy codes** with more flexibility, mainly targeted **to advanced concepts**
- Distributed to 20+ institutions. Now freely available from GitLab (link on IAEA/ONCORE website)

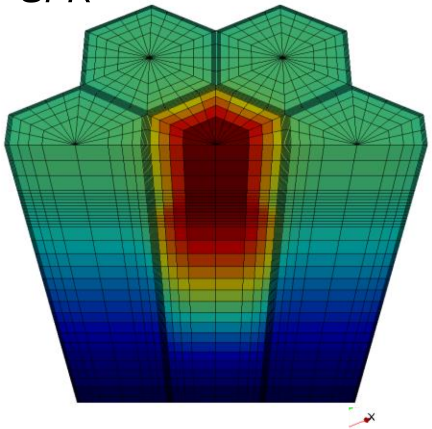


Multi-physics modelling of the MSRE

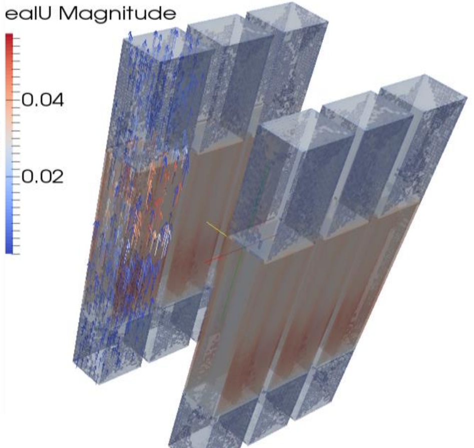
Core flowering in a SFR



Assembly windows in a SFR

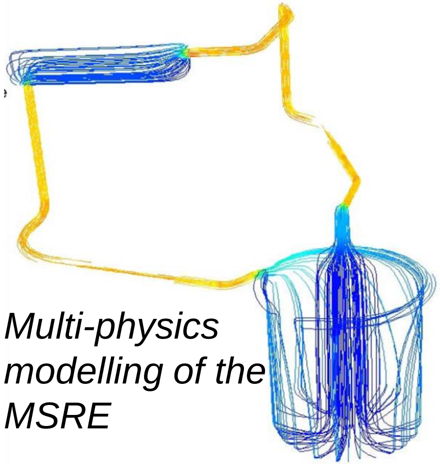


The Argonaut reactor



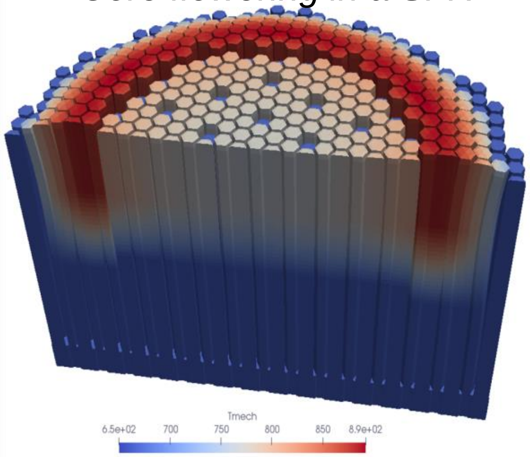
GeN-Foam: Generalized Nuclear Field operation and manipulation

- **Status:**
  - Source code - Stable version with a complete set of functionalities for most applications
  - V&V – Mostly verified. Validation ongoing.
  - Documentation - First version of a doxygen-based documentation + tutorials
- **An extremely flexible code**

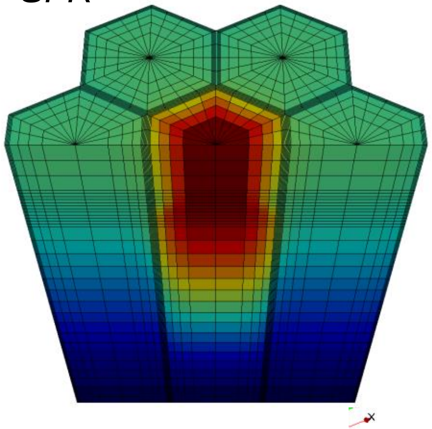


Multi-physics modelling of the MSRE

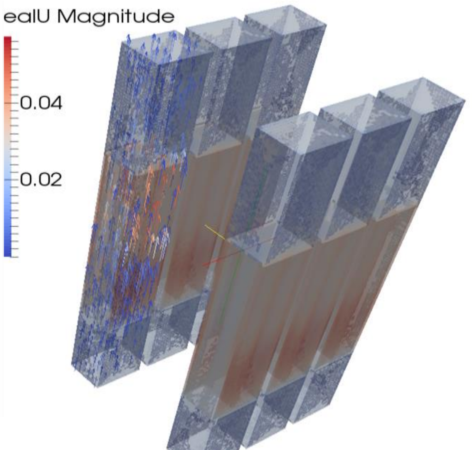
Core flowering in a SFR



Assembly windows in a SFR

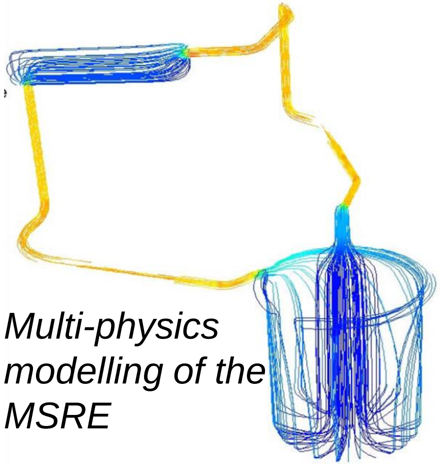


The Argonaut reactor



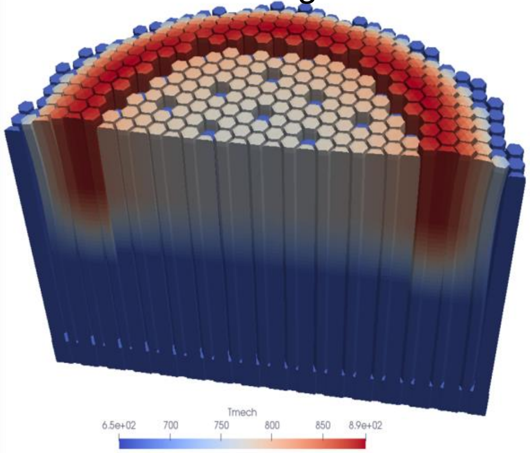
GeN-Foam: Generalized Nuclear Field operation and manipulation

- **Status:**
  - Source code - Stable version with a complete set of functionalities for most applications
  - V&V – Mostly verified. Validation ongoing.
  - Documentation - First version of a doxygen-based documentation + tutorials
- **An extremely flexible code that requires some commitment and sound background both in nuclear engineering and numerical analysis**

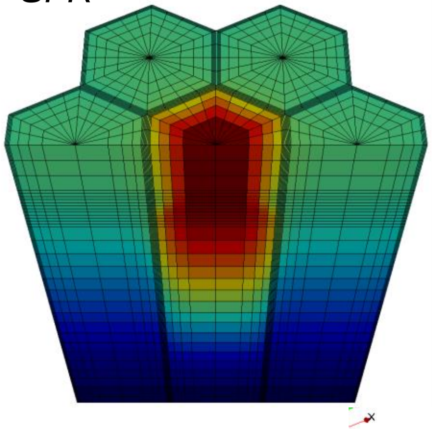


Multi-physics modelling of the MSRE

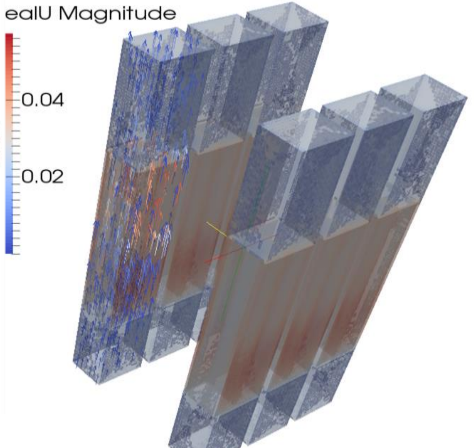
Core flowering in a SFR



Assembly windows in a SFR



The Argonaut reactor



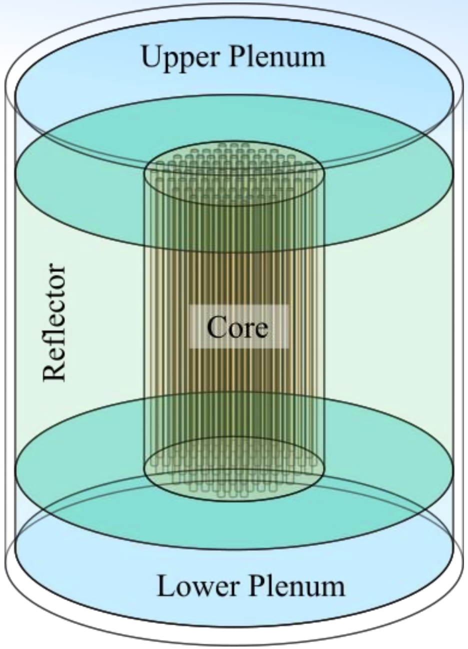
GeN-Foam: V&V status

Brief description	Neutronics	Thermal-hydraulics	Thermal-mechanics	Coupling
Comparison against PARCS for a PWR a mini-core [15]	x (SP3)			
Comparison against Serpent for the CROCUS reactor [15]	x (SP3)			
Comparison against Serpent for the ESFR [17]	x (Diffusion)			
Comparison against Serpent for a PWR mini-core [17]	x (Diffusion)			
Comparison against various codes for the ESFR-SMART design [21]	x (Diffusion)			
Verification against analytic solutions for a simplified MSR [22]	x (Diffusion)	x (1 phase)		x
Verification against the CNRS MSR benchmark [23]	x (Diffusion)	x (1 phase)		x
Comparison against TRACE for the ESFR core [3,18]	x (Diffusion)	x (1 phase)	x	x
Verification using the method of manufactured solutions [6]		x (1-2 phases)		
Validation against the Godiva IV experiment [16]	x (SN)			
Validation against the FFTF LOFWOS Test 13 [4]	x (pk)	x (1 phase)		
Validation against the KNS-3-L22 experiment on sodium boiling [4]		x (1-2 phases)		
Validation against the ISPRA experiment on sodium boiling [4]		x (1-2 phases)		
Validation against the NEA PSBT benchmark on water boiling		x (1-2 phases)		
Validation against CROCUS measurements [19]	x (Diff, SP3, SN)			

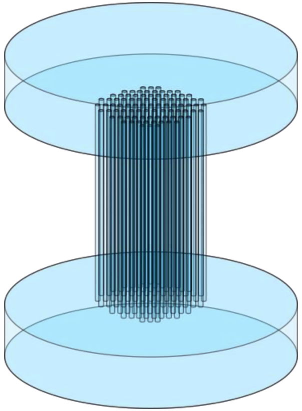
Basics

Let's consider some hypothetical reactor
- Core with coolant channels
- Lower and upper plena
- RPV

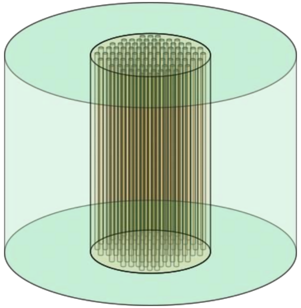
We want to model thermal-hydraulics coupled to 3D kinetics and thermal-mechanics



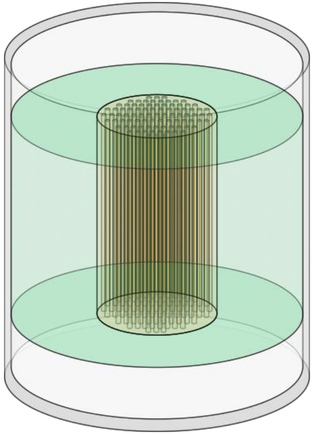
Basics



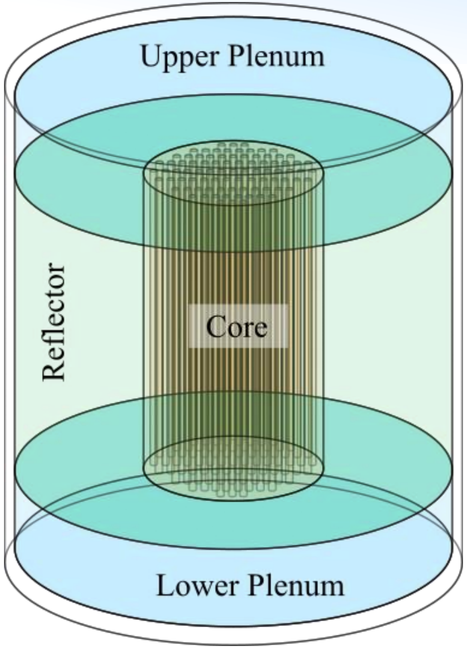
Coolant



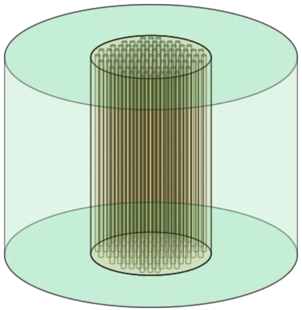
Neutronics



Solid Structures



Basics

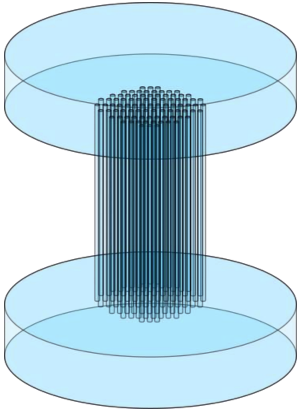


Neutronics

Neutronics **mesh**

Fields:
Fluxes, DN precursors
Cross-sections, *power*

Equations:
neutron transport / diffusion, delayed neutron production/decay/transport

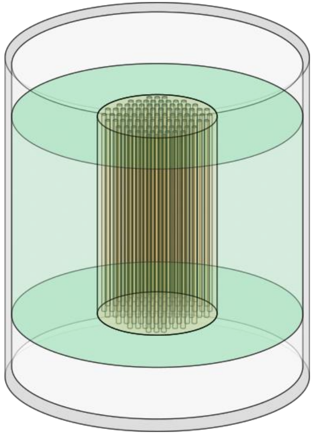


Coolant

Fluid **mesh**

Fields:
Velocity, Pressure, Temperature, thermophysical properties

Equations: RANS (porous?)



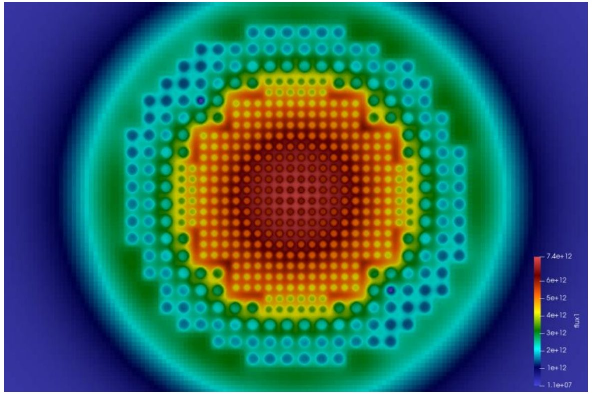
Solid Structures

Thermo-mechanic **mesh**

Fields: Temperature, Displacement, thermophysical properties, *stresses, strains*

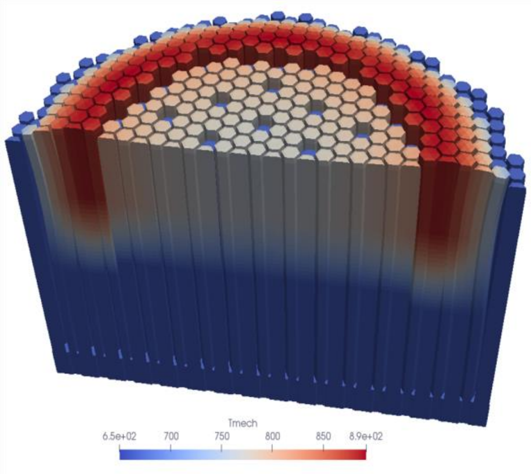
Equations:
Heat conduction (porous?)
Cont. mechanics (porous?)

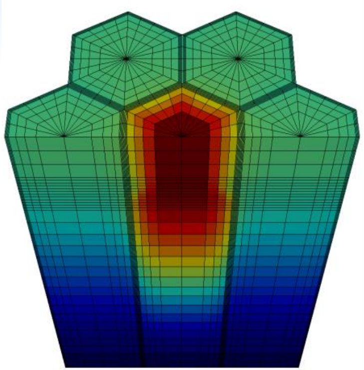
Physics in GeN-Foam

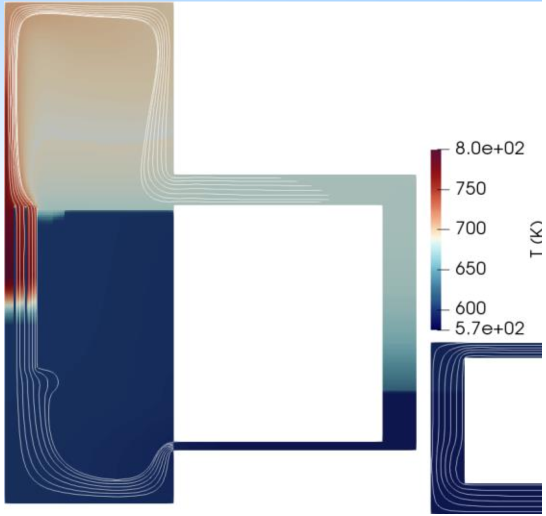


<u>Neutronics</u>

- Diffusion
- Adjoint diffusion
- SP3
- SN
- Point-kinetics
- Precursor transport







<u>Thermal-hydraulics:</u>

- RANS CFD + porous-med
- One and two phase
- Two phase models for sodium and water (not fully validated)

<u>Thermal-mechanics</u>

- Linear elasticity
- BC for multi-material and contact

Physics in GeN-Foam

Source code

master | GeN-Foam / GeN-Foam / classes / | + | Lock | History | Find file | Web I

Merge branch 'develop'
foam-for-nuclear project authored 2 months ago

Name	Last commit
..	
multiphysicsControl	IPorted restructuring of FFSEulerFoam (as of commit 5fd0cfd7fbb32ec7...
neutronics	Merge branch 'develop' after upgrade to OF v2112
thermalHydraulics	Updated to OpenFOAMv2206
thermoMechanics	Upgrade to OpenFOAM v2112

Case folder

master | GeN-Foam / Tutorials / 3D_SmallESFR / rootCase / 0 / | + | Lock | History | Find file | Web I

Updated GeN-Faom to OpenFOAM v2006, which broke some aspects of FFSEulerFoam...
Stefan Radman authored 2 years ago

Name	Last commit
..	
fluidRegion	Updated GeN-Faom to OpenFOAM v2006, which broke some aspec...
neutroRegion	All tutorials have been updated with the exception of the regressio...
thermoMechanicalRegion	All tutorials have been updated with the exception of the regressio...
cellToRegion	All tutorials have been updated with the exception of the regressio...

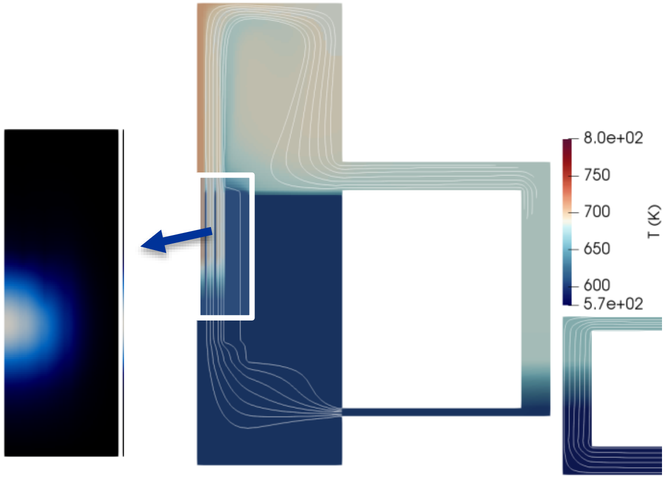
controlDict

```
// Physics to solve

solveFluidMechanics     true;

solveEnergy             true;

solveNeutronics         true;

solveThermalMechanics   true;
```

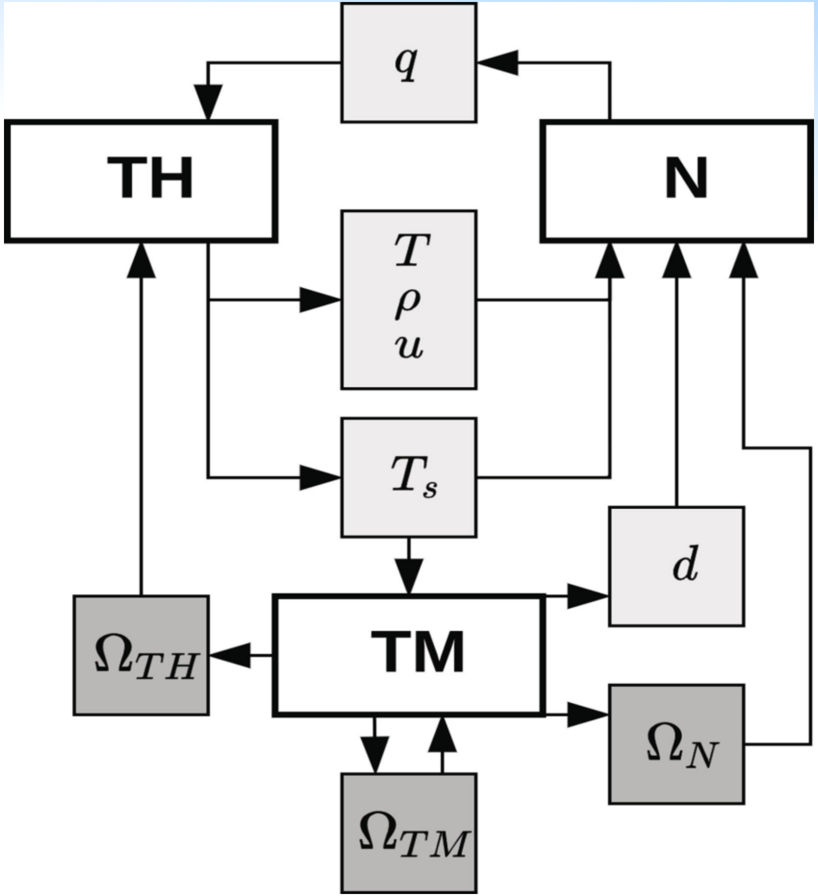
Coupling of physics

- 3 different meshes
  - Different refinements

  - Different regions of the reactor



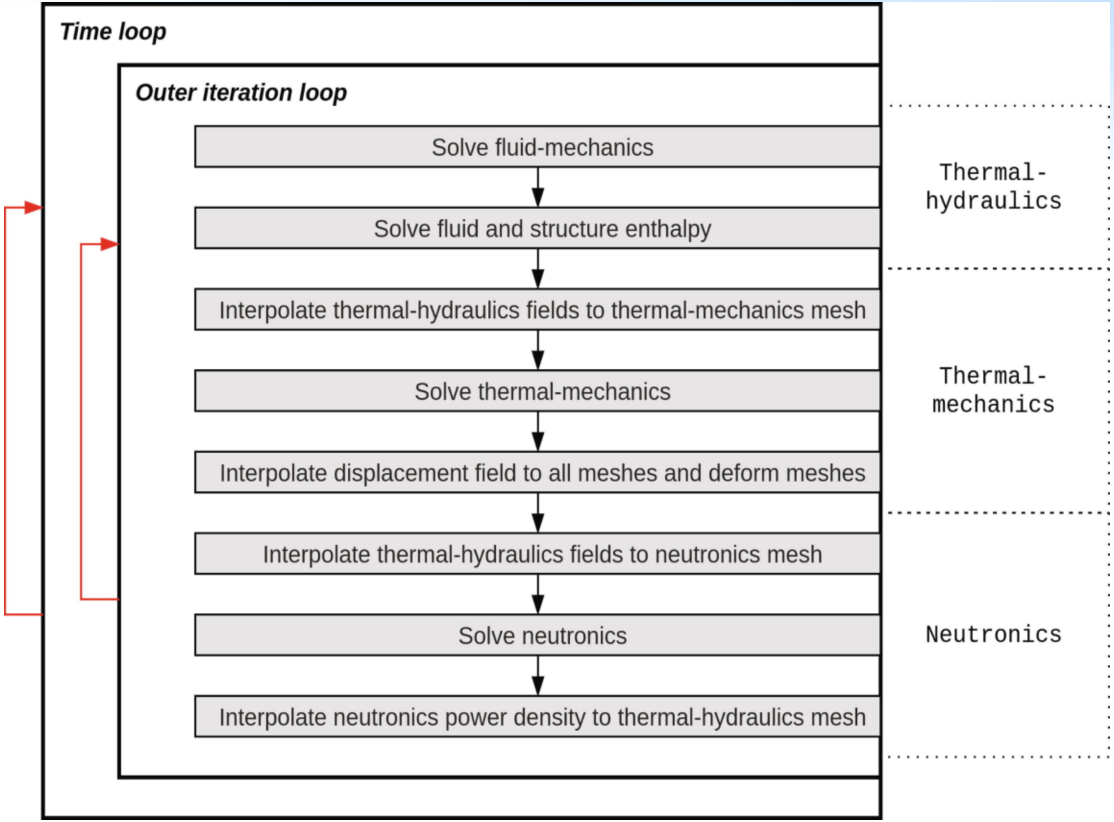
- Mesh-to-mesh projection of coupling fields



Coupling of physics

- Fixed-point iteration
  - Simple
  - Accurate
  - Stable
  - Well-suited for modular / extensible code

- Semi-implicit:
  - Extended PIMPLE loop: pressure-velocity coupling rarely fully implicit in commercial CFD solvers
  - The rest can be iterated till full convergence



Coupling of physics

- Neutronics – energy – thermal-mechanics coupling can be fully resolved on last pimple iteration

```
//- Correct flow regime map
if (solveFM or solveE)
{
    thermalHydraulics.correctModels(solveFM, solveE);
}

//- Solve fluid-mechanics
if (solveFM)
{
    thermalHydraulics.correctFluidMechanics(FMResidual);
    if (!solveE and !solveN and !solveTM)
        Info << endl;
}

//- Solve energy
if (!multiphysics.finalIter())
{
    if (solveE)
    {
        thermalHydraulics.correctEnergy(EResidual);
        if (!solveN and !solveTM)
            Info << endl;
    }
}
```

```
//- Solve energy-neutronics-thermomechanics coupling on last outer iteration
else if (solveE or solveN or solveTM)
{
    scalar couplingResidual = 0.0;
    label couplingIter = 0;
    do
    {
        Info << "Coupling iteration " << couplingIter << endl;

        //- Reset as the thermoMechanics.correct(couplingResidual) and
        //  neutronics.correct(couplingResidual) always max() it against their
        //  solution residual, meaning that with no reset, it will stay stuck
        //  at its max value (likely the one of the first coupling iteration)
        couplingResidual = 0.0;

        if (solveE)
        {
            thermalHydraulics.correctEnergy(couplingResidual);
            if (!solveN and !solveTM)
                Info << endl;
        }
        if (solveTM or solveN)
        {
            #include "correctCouplingFields.H"
        }
        if (solveTM)
        {
            thermoMechanics.interpolateCouplingFields(mechToFluid);
            thermoMechanics.correct(couplingResidual);
            neutronics.deformMesh(mechToNeutro,thermoMechanics.meshDisp());
        }
        if (solveN)
        {
            neutronics.interpolateCouplingFields(neutroToFluid);
            neutronics.correct
            (
                couplingResidual,
                couplingIter
            );
            (*powerDensity) *= 0.0;
            fluidToNeutro.mapTgtToSrc
            (
                neutronics.powerDensity(),
                plusEqOp<scalar>(),
                powerDensity->primitiveFieldRef()
            );
        }
        couplingIter++;
```

Coupling of physics

master

GeN-Foam / Tutorials / 3D_SmallESFR / rootCase / system / fluidRegion / fvSolution

Single-physics parameters

After the last large commit from Stefan (dc0c292d), •••
foam-for-nuclear project authored 11 months ago

fvSolution 1.78 KiB

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  6
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSolution;
}

PIMPLE // a detailed explanation of this dictionary is available in this
       // same fvSolution file of the 1D_boiling tutorial
{
    nCorrectors      2;//pressure-velocity correctors
    nNonOrthogonalCorrectors 0;
    // partialEliminationMode      implicit;
    momentumMode                faceCentered;
}

relaxationFactors
{
    equations
    {
        ".*"            1;
    }
}
```

master

GeN-Foam / Tutorials / 3D_SmallESFR / rootCase / system / **fvSolution**

Tutorial ESFR: added README file, commented controlDict and •••
foam-for-nuclear project authored 2 years ago

Global parameter

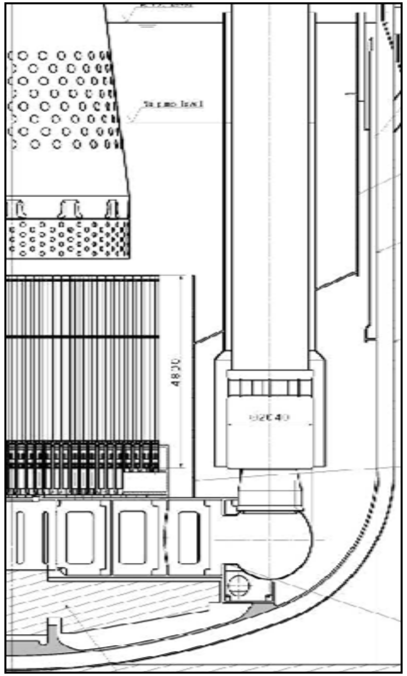
fvSolution 1.43 KiB

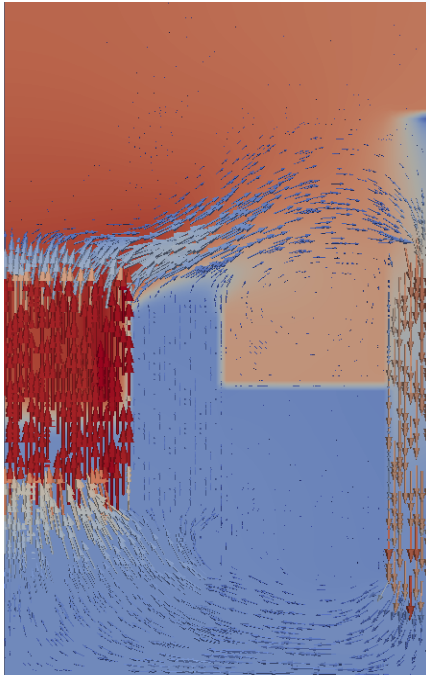
```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //


    nOuterCorrectors        6; // number of energy-pressure-velcity correctors
    tightlyCoupled          false; // tight coupling, at each time step, of
                                   // neutronics, energy and thermal-mechanics.
                                   // The coupling is regulated by the two
                                   // parameters below
    timeStepResidual        0.00005; // max allowed residual at each time step
    maxTimeStepIterations   6; // for transient.
                               // Maximum iterations in the sub-loop between
                               // neutronics, energy and thermal-mechanics.
                               // The sub-loop is performed at the last outer
                               // corrector (see flag above)

// ************************************************************************* //
```

Thermal-hydraulics

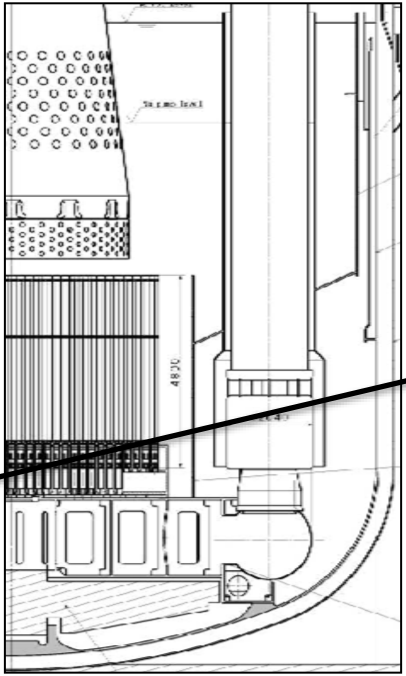
- GeN-Foam was born for full-core and full-primary-circuit safety analyses
- Need for reducing computational footprint w.r.t. RANS models
- In legacy nuclear codes:
  - 1-D system-code approach
  - Sub-channel approach
- In GeN-Foam (and other solvers based on PDE libraries): porous-medium approach
  - Can be based on standard CFD solution algorithms
  - Equivalent to 1-D system codes if restricted to 1-D (essentially, a 3-D version of a system code where interaction with the structure is modelled using drag coefficients and Nusslet numbers)
  - Can reproduce results of sub-channel codes if properly tuned
  - Reverts back to fine-mesh RANS models in clear-fluid regions (plena, pools) -> fully implicit hybrid coarse/fine mesh simulations

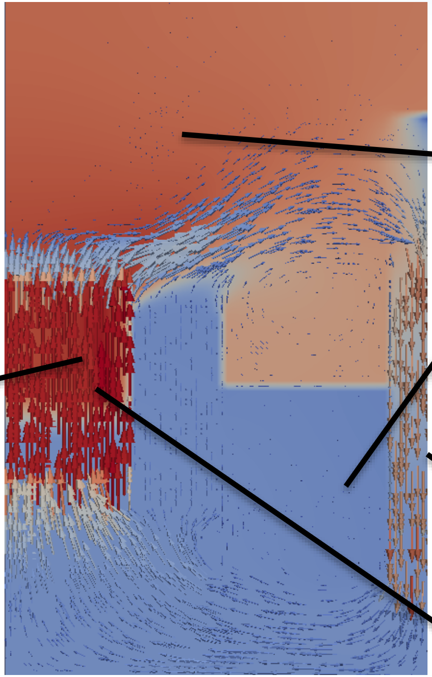
Thermal-hydraulics: combined coarse / fine-mesh





Thermal-hydraulics: combined coarse / fine-mesh





CFD for pools

Some models for sub-scale structures

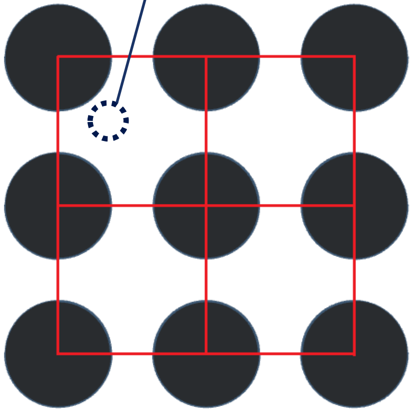
coarse-mesh porous-medium for core and HXs

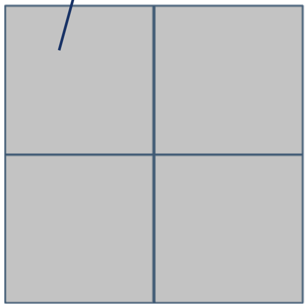
Porous-medium thermal-hydraulics: volume averaging

$$\langle \phi_i^* \rangle_I = \frac{1}{V_i}\int_{\Omega_i} \phi_i^* \, dV$$

$$\langle \phi_i^* \rangle = \frac{1}{V}\int_{\Omega_i} \phi_i^* \, dV$$

$$\frac{\partial}{\partial t}\rho_i^* + \nabla \cdot (\mathbf{u}_i^* \rho_i^*) = 0 \longrightarrow \frac{\partial}{\partial t}(\alpha_i \rho_i) + \nabla \cdot (\alpha_i \mathbf{u}_i \rho_i) = \boxed{\sum_{j \neq i} \frac{1}{V} \int_{\partial \Omega_{ij}} \rho_i^* \left(\mathbf{u}_\partial^* - \mathbf{u}_i^*\right) \cdot \mathbf{n} \, dS}$$





Volume averaging results in:

- Additional variables (phase fraction, tortuosity, etc.);
- Additional source terms that require experimentally-informed closure;

Porous-medium thermal-hydraulics: governing equations

The multi-phase coarse-mesh governing equations (Navier-Stokes and enthalpy) are:

$$\frac{\partial}{\partial t}(\alpha_i \rho_i) + \boldsymbol{\nabla} \cdot (\alpha_i \mathbf{u}_i \rho_i) = -\Gamma_{i\to j}$$

$$\frac{\partial}{\partial t}(\alpha_i \rho_i \mathbf{u}_i) + \boldsymbol{\nabla} \cdot (\alpha_i \rho_i \mathbf{u}_i \otimes \mathbf{u}_i) =$$

$$- \alpha_i \boldsymbol{\nabla} p + \boldsymbol{\nabla} \cdot (\alpha_i \boldsymbol{\sigma}_{d,i}) + \alpha_i \rho_i \mathbf{g} - \mathbf{S}_{\mathbf{u},i\to j}$$

$$\frac{\partial}{\partial t}(\alpha_i \rho_i h_i) + \boldsymbol{\nabla} \cdot (\alpha_i \mathbf{u}_i \rho_i h_i) =$$

$$\boldsymbol{\nabla} \cdot (\alpha_i \kappa_i \boldsymbol{T}_i \cdot \boldsymbol{\nabla} T_i) + \alpha_i \frac{\partial}{\partial t} p + \alpha_i \rho_i \mathbf{u}_i \cdot \mathbf{g} + \alpha_i q_{int,i} - S_{h,i\to j}$$

Porous-medium thermal-hydraulics: governing equations

The multi-phase coarse-mesh governing equations (Navier-Stokes and enthalpy) are:

$$\frac{\partial}{\partial t}(\alpha_i \rho_i) + \boldsymbol{\nabla} \cdot (\alpha_i \mathbf{u}_i \rho_i) = -\Gamma_{i \to j}$$

Mass transfer between phases

$$\frac{\partial}{\partial t}(\alpha_i \rho_i \mathbf{u}_i) + \boldsymbol{\nabla} \cdot (\alpha_i \rho_i \mathbf{u}_i \otimes \mathbf{u}_i) =$$

$$-\alpha_i \boldsymbol{\nabla} p + \boldsymbol{\nabla} \cdot (\alpha_i \boldsymbol{\sigma}_{d,i}) + \alpha_i \rho_i \mathbf{g} - \mathbf{S}_{\mathbf{u},i \to j}$$

Momentum exchange with other phases (or structure)

Volume fraction occupied by the phase

$$\frac{\partial}{\partial t}(\alpha_i \rho_i h_i) + \boldsymbol{\nabla} \cdot (\alpha_i \mathbf{u}_i \rho_i h_i) =$$

$$\boldsymbol{\nabla} \cdot (\alpha_i \kappa_i \boldsymbol{T}_i \cdot \boldsymbol{\nabla} T_i) + \alpha_i \frac{\partial}{\partial t} p + \alpha_i \rho_i \mathbf{u}_i \cdot \mathbf{g} + \alpha_i q_{int,i} - S_{h,i \to j}$$

Energy exchange with other phases (or structure)

Porous-medium thermal-hydraulics: governing equations

The multi-phase coarse-mesh governing equations (Navier-Stokes and enthalpy) are:

$$\frac{\partial}{\partial t}(\alpha_i\rho_i) + \boldsymbol{\nabla}\cdot(\alpha_i\mathbf{u}_i\rho_i) = -\Gamma_{i\to j}$$

Mass transfer between phases

Volume fraction occupied by the phase

$$\frac{\partial}{\partial t}(\alpha_i\rho_i\mathbf{u}_i) + \boldsymbol{\nabla}\cdot(\alpha_i\rho_i\mathbf{u}_i\otimes\mathbf{u}_i) = - \alpha_i\boldsymbol{\nabla}p + \boldsymbol{\nabla}\cdot(\alpha_i\boldsymbol{\sigma}_{d,i}) + \alpha_i\rho_i\mathbf{g} - \mathbf{S}_{\mathbf{u},i\to j}$$

Momentum exchange with other phases (or structure)

$$\frac{\partial}{\partial t}(\alpha_i\rho_i h_i) + \boldsymbol{\nabla}\cdot(\alpha_i\mathbf{u}_i\rho_i h_i) = \boldsymbol{\nabla}\cdot(\alpha_i\kappa_i\boldsymbol{T}_i\cdot\boldsymbol{\nabla}T_i) + \alpha_i\frac{\partial}{\partial t}p + \alpha_i\rho_i\mathbf{u}_i\cdot\mathbf{g} + \alpha_i q_{int,i} - S_{h,i\to j}$$

Energy exchange with other phases (or structure)

These reduce to traditional CFD approaches in clear fluid regions and a system-code-like approach in 1-D regions (multiple scales).

Porous-medium thermal-hydraulics: governing equations

In one phase, with some changes in notation:

$$\boldsymbol{\nabla} \cdot \boldsymbol{u} = 0$$

Volume fraction occupied by the phase = porosity

$$\frac{\partial(\chi\rho\boldsymbol{u})}{\partial t} + \boldsymbol{\nabla} \cdot (\chi\rho\boldsymbol{u} \otimes \boldsymbol{u}) = \boldsymbol{\nabla} \cdot (\mu_t \boldsymbol{\nabla}\boldsymbol{u}) - \boldsymbol{\nabla}(\chi p) + \chi\boldsymbol{F}_{\boldsymbol{g}} + \boxed{\chi\boldsymbol{F}_{\boldsymbol{ss}}}$$

Momentum exchange with the sub-scale structure

$$\frac{\partial(\chi\rho e)}{\partial t} + \boldsymbol{\nabla} \cdot \left(\chi\rho\boldsymbol{u}\left(e + \frac{p}{\rho}\right)\right) = \boldsymbol{\nabla} \cdot (\chi k_t \boldsymbol{\nabla} T) + \boldsymbol{F}_{\boldsymbol{ss}} \cdot \boldsymbol{u} + \boxed{\chi\dot{Q}}$$

Energy exchange with the sub-scale structure

Porous-medium thermal-hydraulics: Sub-scale structures – momentum exchange

$$\frac{\partial(\chi\rho\boldsymbol{u})}{\partial t} + \boldsymbol{\nabla}\cdot(\chi\rho\boldsymbol{u}\otimes\boldsymbol{u}) = \boldsymbol{\nabla}\cdot(\mu_t\boldsymbol{\nabla}\boldsymbol{u}) - \boldsymbol{\nabla}(\chi p) + \chi\boldsymbol{F_g} + \chi\boldsymbol{F_{ss}}$$

$$\boldsymbol{F_{ss}} = \boldsymbol{\kappa}(\boldsymbol{u_D})\cdot\boldsymbol{u_D}$$

$$\kappa(u_D)_{ii} = \frac{f_{D,i}\rho u_{D,i}}{2D_h\gamma^2}$$

In 1-D, steady state

$$\frac{\Delta p}{L} = \frac{\partial p}{\partial x} = F_{ss,x} = 0.5 f_D \rho v^2 \frac{1}{D}$$

$$\Delta p = 0.5 f_D \rho v^2 \frac{L}{D}$$

Darcy friction factor

Porous-medium thermal-hydraulics: Sub-scale structures – energy exchange

$$\frac{\partial(\chi\rho e)}{\partial t} + \boldsymbol{\nabla}\cdot\left(\chi\rho\boldsymbol{u}\left(e+\frac{p}{\rho}\right)\right) = \boldsymbol{\nabla}\cdot(\chi k_t \boldsymbol{\nabla} T) + \boldsymbol{F_{ss}}\cdot\boldsymbol{u} + \chi\dot{Q}$$

$$\dot{Q}_{ss} \propto h(T_{SS} - T)$$

Temperature of subscale structure

Heat transfer coefficient, h(Nu)

Nu from correlations

$h(T_{SS} - T)$ in W/m^2

Multiply by volumetric area

$$\dot{Q}_{ss} = A_V h(T_{SS} - T)$$

Porous-medium thermal-hydraulics: Sub-scale structures – energy exchange

develop | GeN-Foam / Tutorials / 3D_SmallESFR / rootCase / constant / fluidRegion / phaseProperties

First draft of user manual
foam-for-nuclear project authored 1 month ago

phaseProperties 7.04 KiB

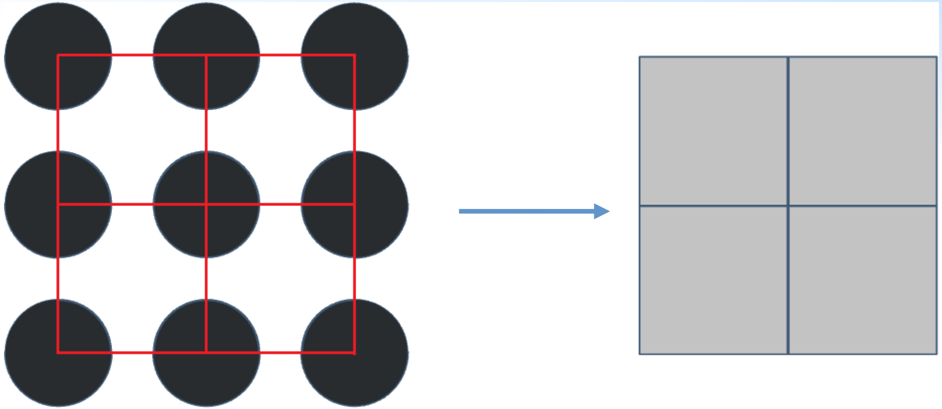
```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  6
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
```

```

regimeMapModels
{
    "lamTurb"
    {
        type                oneParameter;
        parameter           "Re";
        regimeBounds
        {
            "laminar"       (0      1000);  //- 0 is automatically extended
                                            //  to -inf
            "turbulent"     (2300   2301);  //- 2031 is automatically extended
                                            //  to +inf
        }
    }
}

```

```

// ------------------------------------------------------------------------- //
// --- REGIME PHYSICS FOR EACH REGIME ------------------------------------- //
// ------------------------------------------------------------------------- //

physicsModels
{
    dragModels
    {
        "diagrid:axialReflector:radialReflector:follower:controlRod:innerCore:outerCore"
        {
            type    ReynoldsPower;
            coeff   0.687;
            exp     -0.25;
        }
    }

    heatTransferModels
    {
        "diagrid:axialReflector:radialReflector:follower:controlRod:innerCore:outerCore"
        {
            type        byRegime;
            regimeMap   "lamTurb";

            //- List of subdicts specifying a heatTransferModel for each regime
            //  in the lamTurb regimeMap
            "laminar"
            {
                // Nu = const + coeff * Re^expRe * Pr^expPr
                type    NusseltReynoldsPrandtlPower;
                const   4;
                coeff   0;
                expRe   0;
                expPr   0;
            }
            "turbulent"
            {
```

Porous-medium thermal-hydraulics: Sub-scale structures – the structures themselves

$$\dot{Q}_{SS} = A_V h(T_{SS} - T)$$



At minimum, 0-D

$$\rho_{SS} c_{p,SS} \frac{\partial T_{SS}}{\partial t} = A_V h(T - T_{SS})$$

Or 0-D with (coarse-mesh) thermal diffusivity

$$\rho_{SS} c_{p,SS} \frac{\partial T_{SS}}{\partial t} = \nabla \cdot (\gamma \boldsymbol{k}_{SS} \nabla T) + A_V h(T - T_{SS})$$

But not always enough…

Porous-medium thermal-hydraulics: Sub-scale structures – the structures themselves

- In GeN-Foam we have passive structures…
  - Modelled as in the previous slide
  - Can be used for instance to model assembly wrappers, reflectors, diagrids, etc.

- … and power models
  - More complex models
  - Can be used together with a passive structure
  - Can be used to model nuclear fuel, electrically heated rods, heat exchangers, fixed-temperature structures, fixed-power structures, etc.

- For example, the nuclearFuelPin power model takes the power density from neutronics (or from a dictionary, if not solving for neutronics); solves, in each cell, a 1-D heat conduction problem in fuel, gap, and cladding; and gives back to the fluid equation the surface temperature of the cladding (which represents Tss in our equations).

Porous-medium thermal-hydraulics

master

GeN-Foam / Tutorials / 2D_MSFR / rootCase / constant / fluidRegion / phaseProperties

Changed powerModels from constantPower and constantTemperature •••

foam-for-nuclear project authored 9 months ago

phaseProperties 2.77 KiB

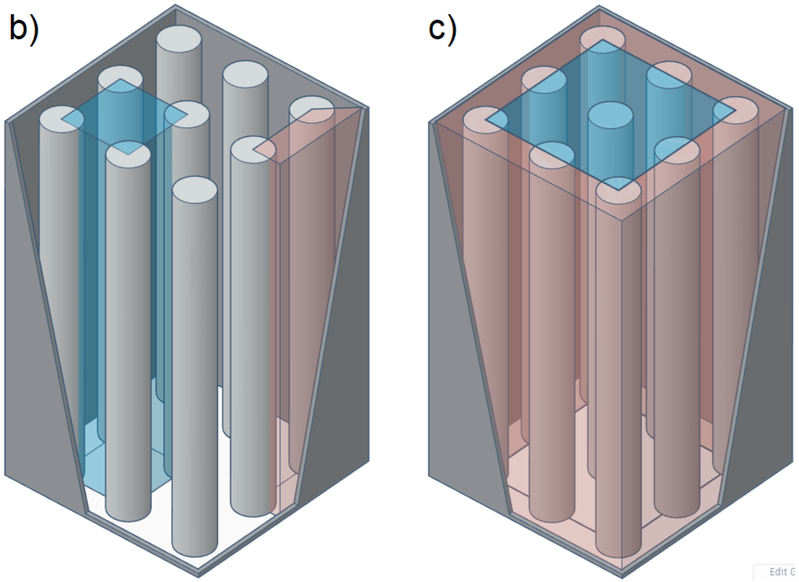
```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  6
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
```

```
thermalHydraulicsType    "onePhase";

// ------------------------------------------------------------------------- //
// --- STRUCTURES PROPERTIES ----------------------------------------------- //
// ------------------------------------------------------------------------- //

structureProperties
{
    "intermed:main_fd"
    {
        volumeFraction      0;
        Dh                  1;
    }

    "hx"
    {
        volumeFraction      0.6;
        Dh                  0.01;

        powerModel
        {
            type            fixedTemperature;
            volumetricArea  200;
            T               900;
        }
```

```
    "innerCore:outerCore"
    {
        volumeFraction      0.718520968;
        Dh                  0.00365;

        powerModel // power production model for the sub-scale structure
        {
            type                nuclearFuelPin;

            // The volumetricArea keyword is now deprecated for the
            // nuclearFuelPin and heatedPin powerModels, as it can be shown
            // that by averaging a cylindrical pin (or a bundle of pins) over
            // a volume of any shape, the interfacialArea and volumeFraction
            // of the resulting porous pin structure are not independent, yet
            // are tied by volumetricArea = 2*volumeFraction/outerPinRadius
            // volumetricArea      267.855;
            powerDensity        0;  //- fields on disk have priority, if they
                                    //  are not found, this value is used
            fuelInnerRadius     0.0012;
            fuelOuterRadius     0.004715;
            cladInnerRadius     0.004865;
            cladOuterRadius     0.005365;
            fuelMeshSize        30;
            cladMeshSize        5;
            fuelRho             10480;
            fuelCp              250;
            cladRho             7500;
            cladCp              500;
            gapH                3000;
            fuelK               3;
            cladK               20;
            fuelT               668;
            cladT               668;
        }

        passiveProperties // these are the properties of the metallic wrappers
        {
            volumetricArea  5;
            rhoCp           4.8e6;
            T               668;
        }
```

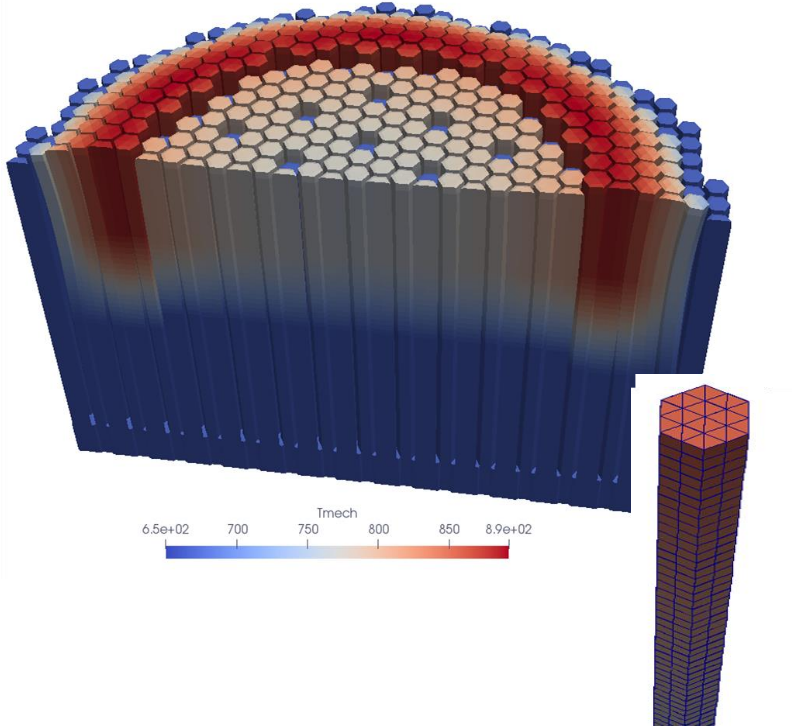
Porous-medium thermal-hydraulics: possibility to mimic sub-channel simulations

One can assign different properties to different regions to replicate results of sub-channel codes

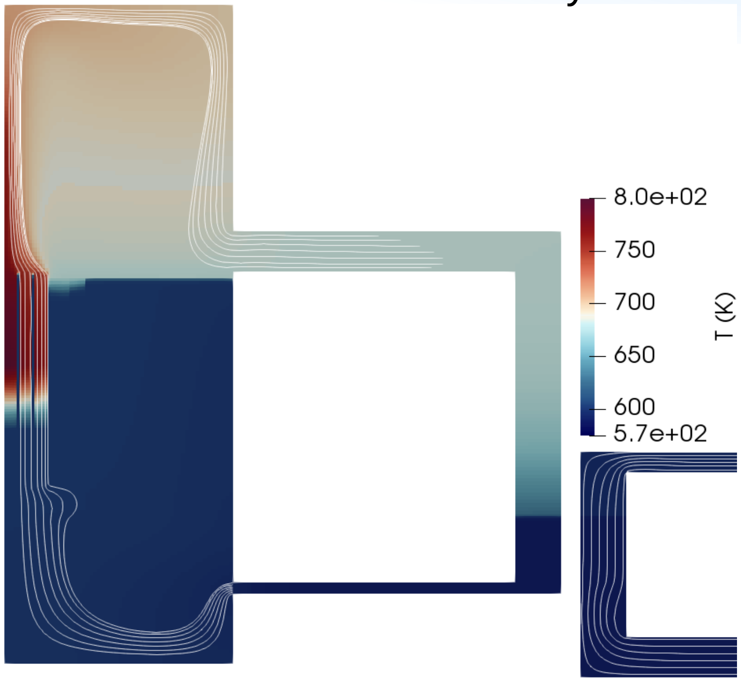


Thermal-hydraulics in GeN-Foam - examples

3-D coarse mesh simulation of a SFR core



2-D combined coarse/fine mesh simulation of the Fast Flux Test Facility



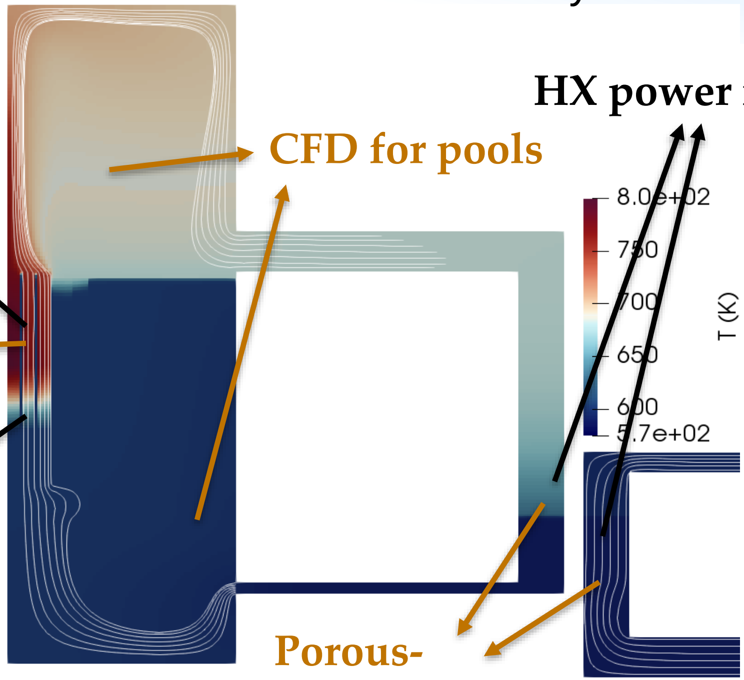
Thermal-hydraulics in GeN-Foam - examples

2-D combined coarse/fine mesh simulation of the Fast Flux Test Facility

Structures in core:
- One passive for wrappers
- One active for fuel

Porous-medium for core and surrounding structures

One structure for diagrid, whose temperature can be used to calculate feedback coefficients



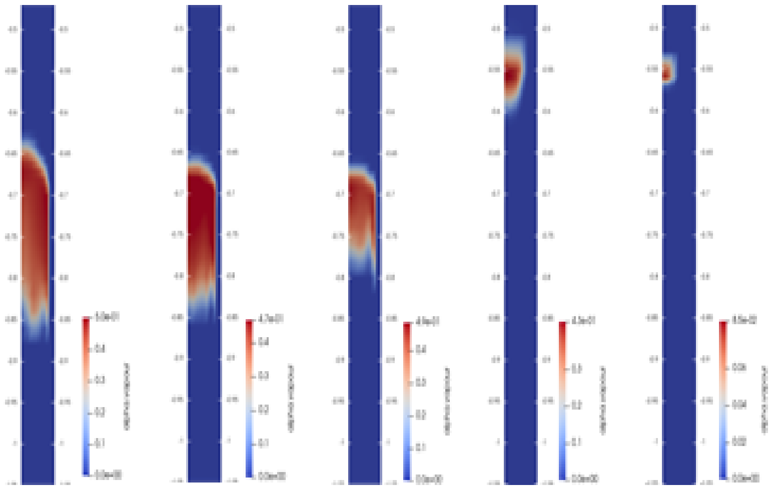
CFD for pools

HX power model

Porous-medium for HX

Two-phase flow

- Same approach as for single-phase thermal-hydraulics (porous-medium with sub-scale structure)
- Beyond the scope of this lecture. Further info in the EPFL PhD thesis of Stefan Radman



2-D coarse mesh simulation of a SFR assembly with windows

Neutronics

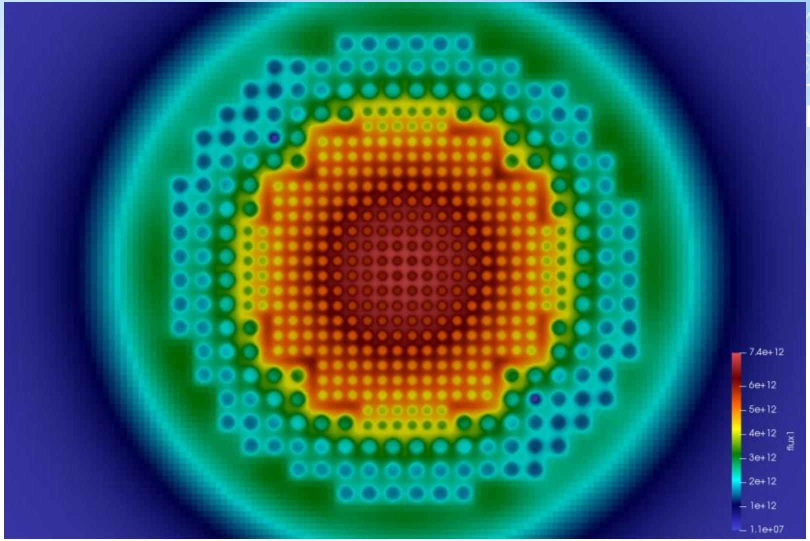
- Models for fluxes/power
  - Diffusion
  - Adjoint diffusion
  - SP3
  - Discrete ordinates (only steady-state)
  - Point-kinetics

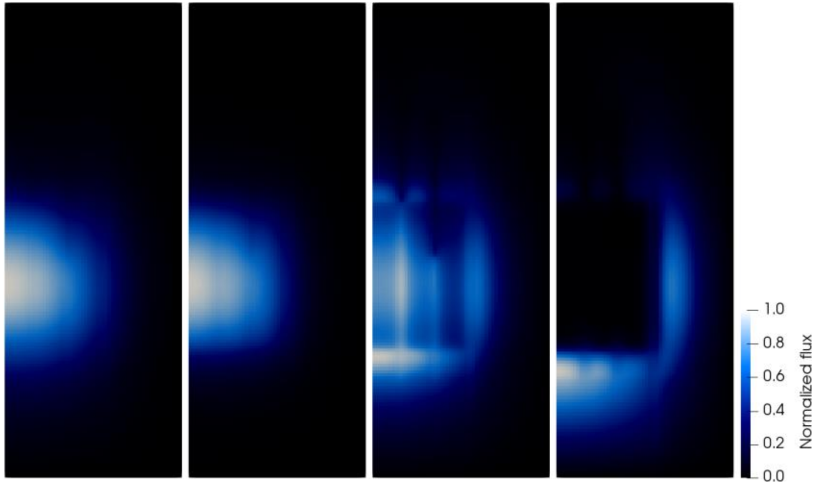
```
fvm::ddt(IV,flux_i])- fvm::laplacian(D,flux_i])= S
```

- Models for precursors
  - Standard balance
  - Precursor transport for MSRs

```
    fvm::ddt(prec_i)
+   fvm::Sp(lambda[precI], prec_i)
-   neutroSource_/keff_*Beta_i
+   fvm::div(phi, prec_i)
-   fvm::laplacian(diffCoeff_, precStar_i)
```

- Eigenvalue or time-dependent
- Multi-group in energy



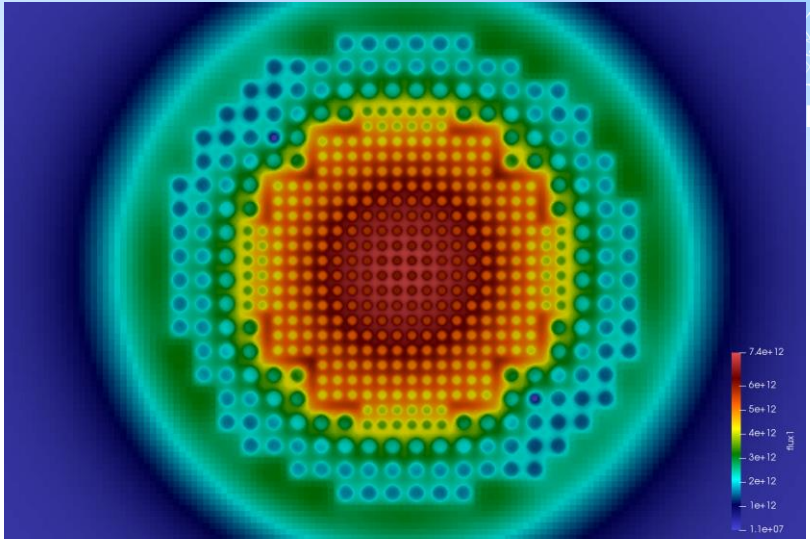


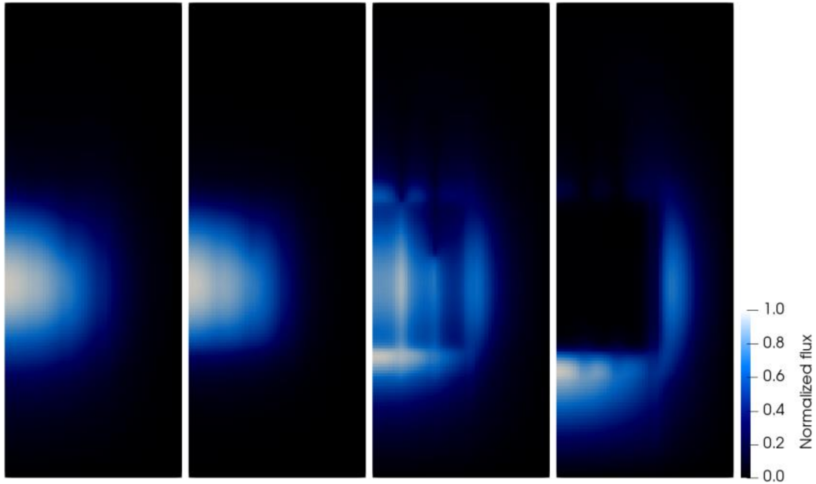
Neutronics

- Dimensionality
  - 1D, 2D, 3D
  - 1D and 2D can be obtained using the empty or wedge boundary condition
  - Exceptions:
    - Point kinetics will adapt…
    - Discrete ordinates: periodic obtained using cyclic BC. Wedge and symmetry won't work.

- Boundary conditions
  - Usual fixed value and zero gradient available
  - Additional albedo BC for diffusion and SP3
  - For discrete ordinates:
    - Specific inlet-outlet BC to model void
    - Dedicated albedo and symmetry under development

- Discretization schemes
  - Gauss harmonic recommended for diffusion and SP3
  - Upwind necessary for discrete ordinates





Neutronics

IAEA

master

GeN-Foam / Tutorials / 2D_MSFR / rootCase / system / controlDict

Restored old mesh.
Peter German authored 1 year ago

controlDict 2.04 KiB

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

// Global simulation options
// This is the crucial flag for MSR simulations. Thanks to this: the power
// calculated by the neutronics will be released directly in the coolant; the
// delayed neutron precursors will be moved according to the collant velocity;
// the fuel temperature will be set equal to the coolant temperature.

liquidFuel                    true;
```

master

GeN-Foam / Tutorials / Godiva_SN / constant / neutroRegion / neutronicsProperties

All tutorials have been updated with the exception of the regression test.... •••
Stefan Radman authored 2 years ago

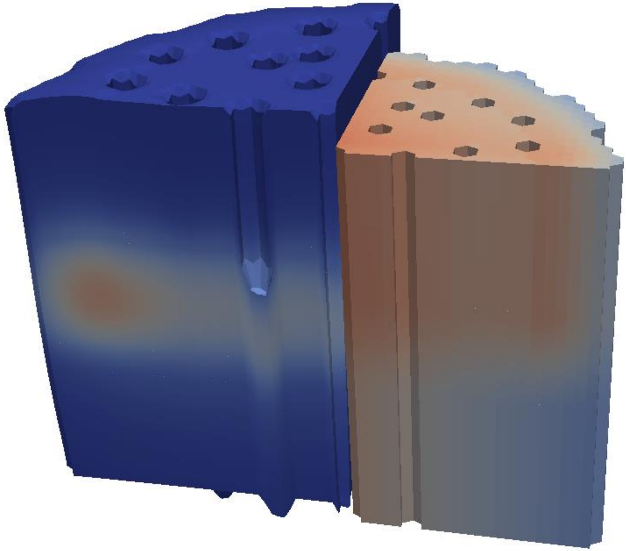
neutronicsProperties 928 bytes

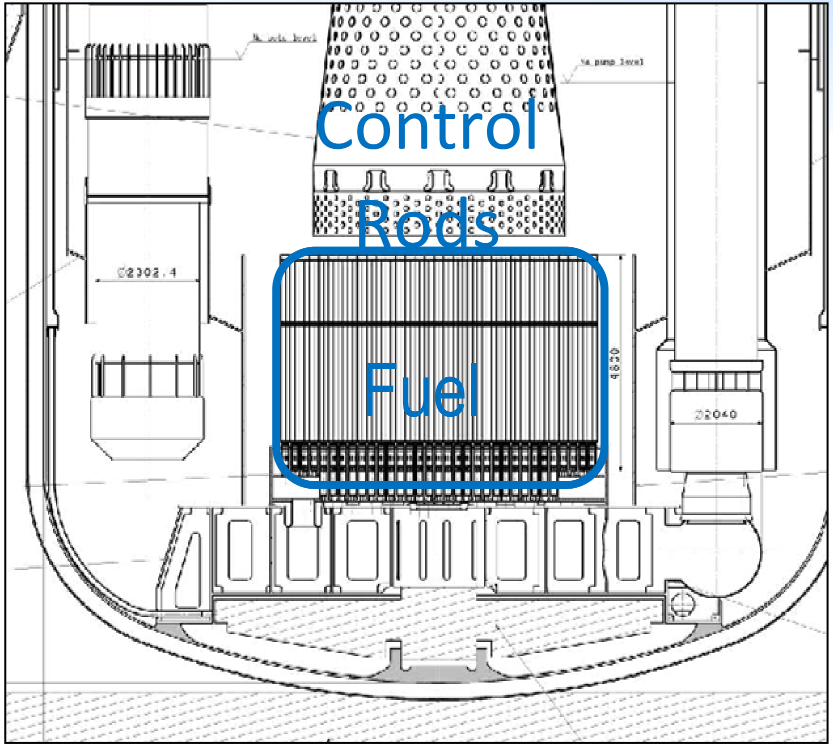
```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      neutronicsProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

model                                   SNNeutronics;

eigenvalueNeutronics    true;
```

Thermal-mechanics (and mesh deformation)

Fuel and CR driveline expansion based on

$$\boldsymbol{v_f} \cdot \nabla D_f = \alpha_{f/c}\left(T_{f/c} - T_{f/c,ref}\right)$$



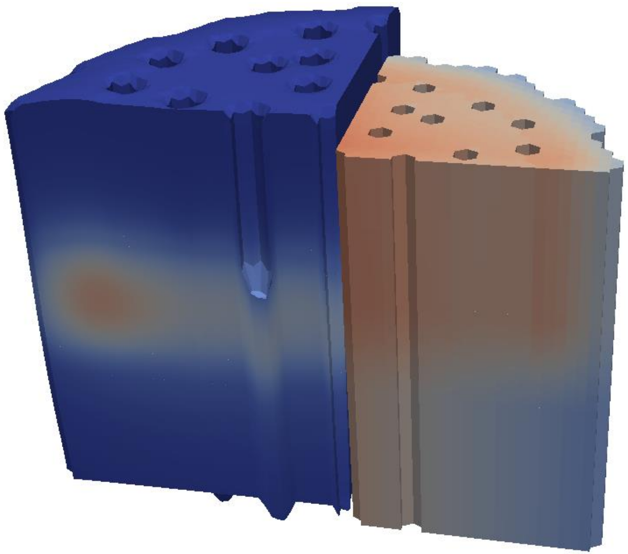


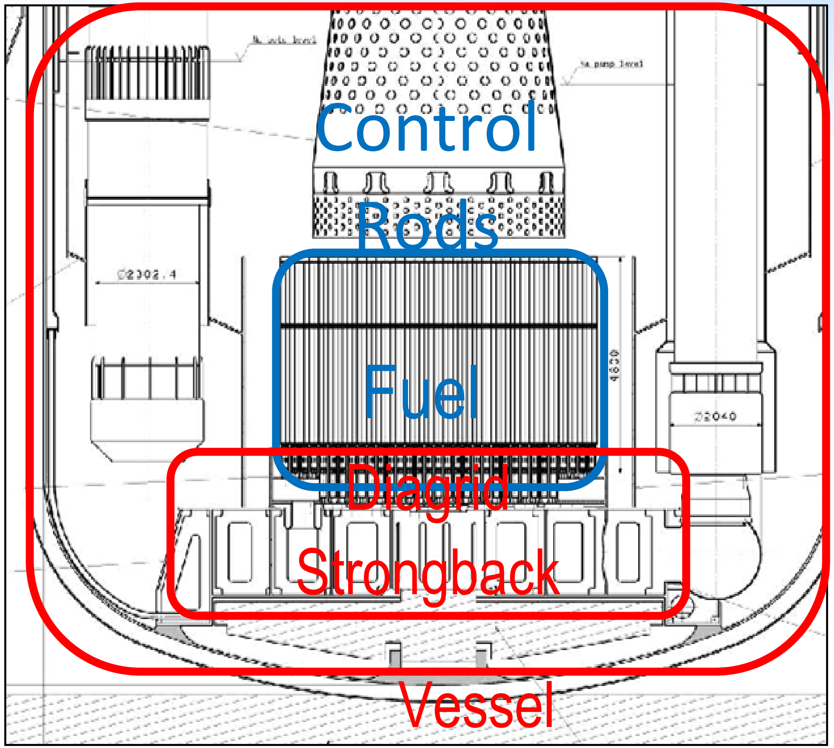
Thermal-mechanics (and mesh deformation)

Fuel and CR driveline expansion based on

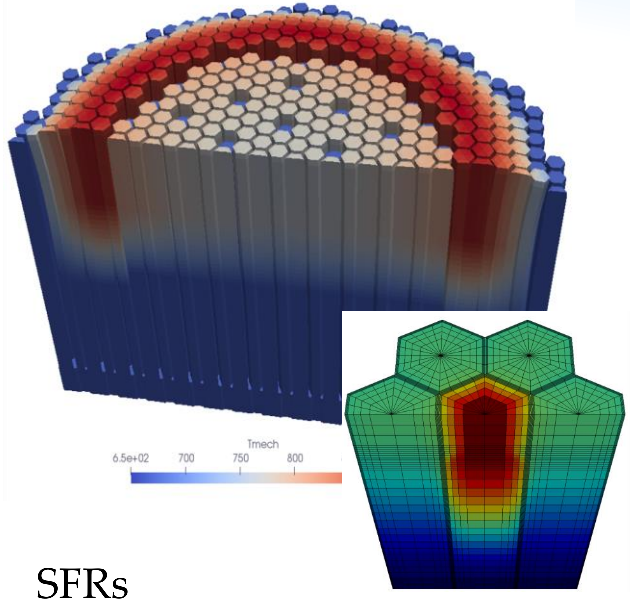
$$\boldsymbol{v}_f \cdot \nabla D_f = \alpha_{f/c}\left(T_{f/c} - T_{f/c,ref}\right)$$

Thermo-elastic solver for other structures





GeN-Foam: examples

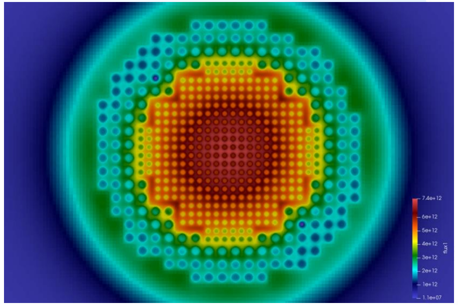


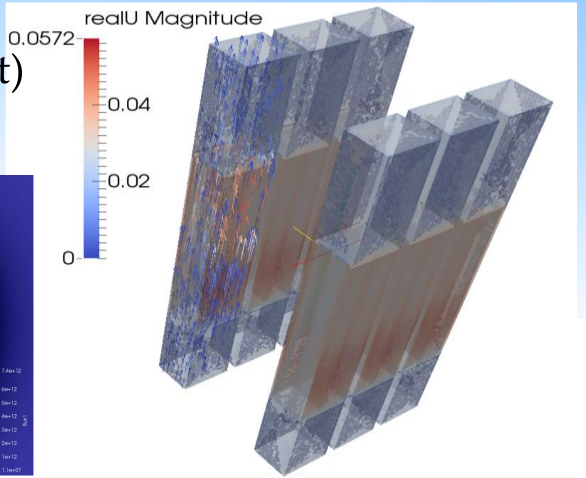
<u>SFRs</u>

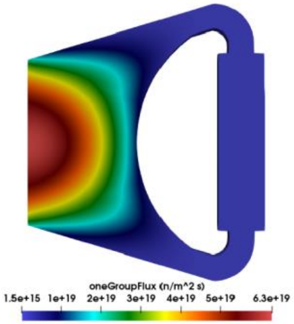
- Multi-dimensional boiling
- Coupling of pools and core
- Direct simulation of deformations

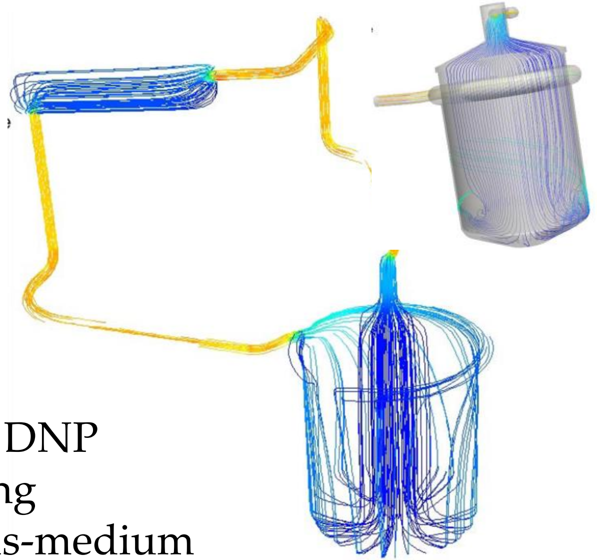
<u>Experimental reactors</u>
(ex, CROCUS, Argounaut)

- Geometric flexibility









<u>MSRs</u>

- Transport of DNP
- Tight coupling
- CFD + porous-medium

GeN-Foam: what else can be modeled?

- LWRs
  - TRACE boiling models implemented and tested
  - Under validation (PSBT)
- HTRs and FHRs
  - Only needs sub-scale model for temperature in pebbles or graphite blocks
- Micro reactors
  - Mainly needs modelling skills
- Heat pipes
  - Under development
- …

GeN-Foam: what else can be modeled?

- LWRs
  - TRACE boiling models implemented and tested
  - Under validation (PSBT)
- HTRs and FHRs
  - Only needs sub-scale model for temperature in pebbles or graphite blocks
- Micro reactors
  - Mainly needs modelling skills
- Heat pipes
  - Under development
- … the limit is your imagination :)

GeN-Foam: Usability

- Complex solver (multi-physics, general finite-volume methodologies on unstructured meshes, linux, …)
  - A background on CFD calculations has been observed to greatly reduce the initial barrier
  - Familiarity with OpenFOAM is necessary
- Somewhat limited documentation
  - Users must be familiar with what they are modelling
- Flexible solver
  - Unstructured meshes, several existing sub-solvers, possibility of tailoring
- Particularly suitable for PhD students and researchers that wish to experiment on methods, address particularly complex problems, or investigate non-traditional reactors
- An expanded documentation and set of tutorials have recently made it possible to use GeN-Foam in the frame of shorter projects such as Master Thesis, as well as a tool for education and training.

Computational requirements

- CPU cores
  - Rule of thumb: 30’000 mesh cells per CPU core
  - CFD
    - 2D RANS-> several hundred thousand cells -> 10 CPU cores
    - 3D RANS -> several hundred millions cells -> 5000 CPU cores
  - Coarse-mesh thermal-hydraulics and neutron diffusion
    - Full-core models -> few hundred thousand to few million cells -> workstations or laptops

- Runtime
  - Steady-state simulations on the optimal number of CPU cores: several minutes to several hours
  - Long-running time-dependent problems: up to a week
  - In some specific applications, such as detailed containment simulations: up to a month

- Memory requirements
  - Single-phase RANS CFD simulation -> order of 10 fields -> 1 GB of memory per million cells
  - 3D discrete ordinates neutron transport -> several thousand solution fields -> 200 GB of memory per million cells

Resources

- Publications

  - C. Fiorina and K. Mikityuk. Application of the new GeN-Foam multi-physics solver to the European Sodium Fast Reactor and verification against available codes. In ICAPP 2015 Conference, Nice, France, 2015.
  - Carlo Fiorina, Ivor Clifford, Manuele Aufiero, and Konstantin Mikityuk. Gen-foam: a novel openfoam® based multi-physics solver for 2d/3d transient analysis of nuclear reactors. Nuclear Engineering and Design, 294:24–37, 2015.
  - Carlo Fiorina, Nordine Kerkar, Konstantin Mikityuk, Pablo Rubiolo, and Andreas Pautz. Development and verification of the neutron diffusion solver for the gen-foam multi-physics platform. Annals of Nuclear Energy, 96:212–222, 2016.
  - Carlo Fiorina, Mathieu Hursin, and Andreas Pautz. Extension of the gen-foam neutronic solver to sp3 analysis and application to the crocus experimental reactor. Annals of Nuclear Energy, 101:419–428, 2017.
  - C. Fiorina, S. Radman, M.-Z. Koc, and A. Pautz. Detailed modelling of the expansion reactivity feedback in fast reactors using OpenFoam. In International Conference on Mathematics and Computational Methods Applied to Nuclear Science and Engineering, M and C 2019, 2019.
  - German, Peter, Ragusa, Jean C., and Fiorina, Carlo. Application of multiphysics model order reduction to doppler/neutronic feedback. EPJ Nuclear Sci. Technol., 5:17, 2019.
  - S. Radman, C. Fiorina, K. Mikityuk, and A. Pautz. A coarse-mesh methodology for modelling of single-phase thermal-hydraulics of ESFR innovative assembly design. Nuclear Engineering and Design, 355, 2019.
  - Stefan Radman, Carlo Fiorina, and Andreas Pautz. Development of a novel two-phase flow solver for nuclear reactor analysis: algorithms, verification and implementation in openfoam. Nuclear Engineering and Design, 379:111178, 2021.
  - Stefan Radman, Carlo Fiorina, and Andreas Pautz. Development of a novel two-phase flow solver for nuclear reactor analysis: Validation against sodium boiling experiments. Nuclear Engineering and Design, 384:111422, 2021.

- Documentation and source code
  - https://foam-for-nuclear.gitlab.io/GeN-Foam/index.html
  - https://gitlab.com/foam-for-nuclear/GeN-Foam/-/tree/master/

- Forum
  - https://foam-for-nuclear.org/phpBB/viewforum.php?f=6&sid=476fa69210b09c168ade3099f5a8c100

Thank you!

Contact: ONCORE@iaea.org

Course Enrolment : Multi-physics modelling and simulation of nuclear reactors using OpenFOAM
ONCORE: Open-source Nuclear Codes for Reactor Analysis