

FROM THE DEEP SEA TO THE ATMOSPHERE

Hand-on with HydrothermalFoam

Lars Rüpke and Zhikui Guo

lruerpke@geomar.de & zguo@geomar.de

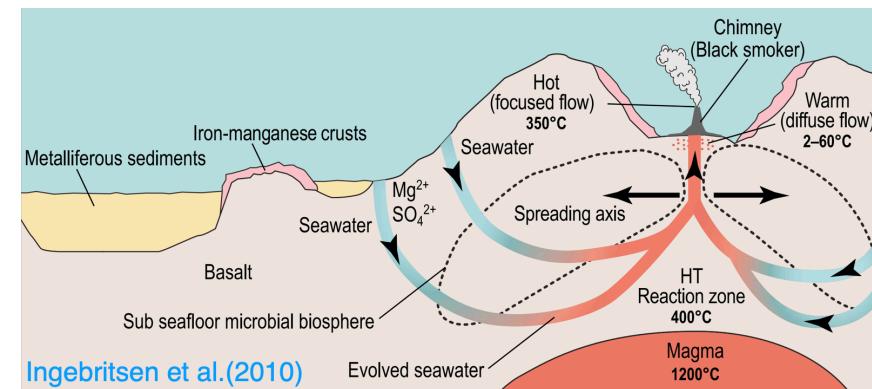
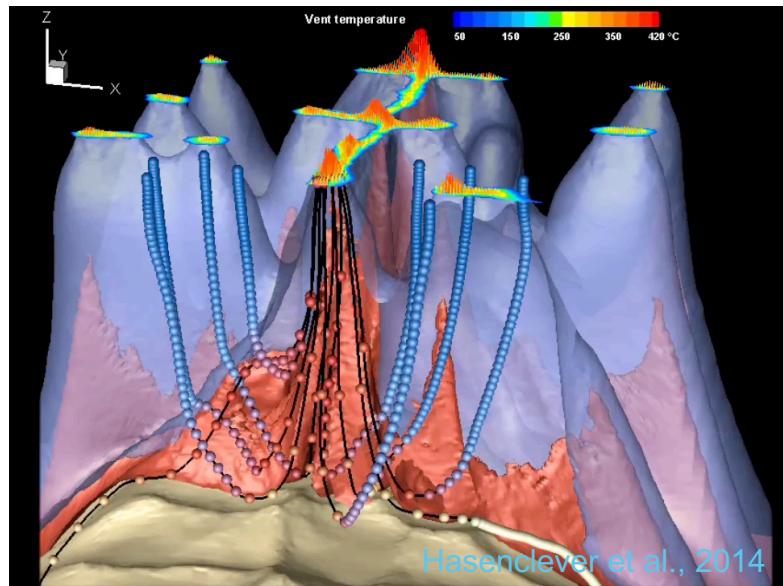
HELMHOLTZ

RESEARCH FOR GRAND CHALLENGES



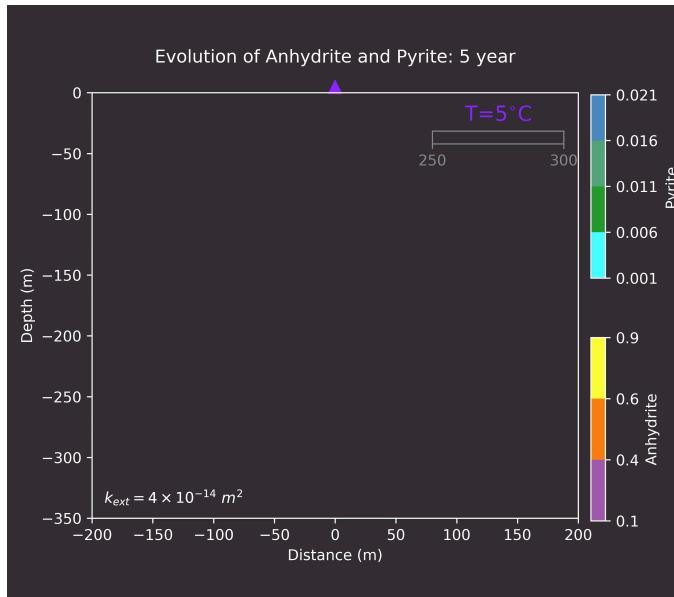
Why use numerical models?

Move from cartoons to physical models

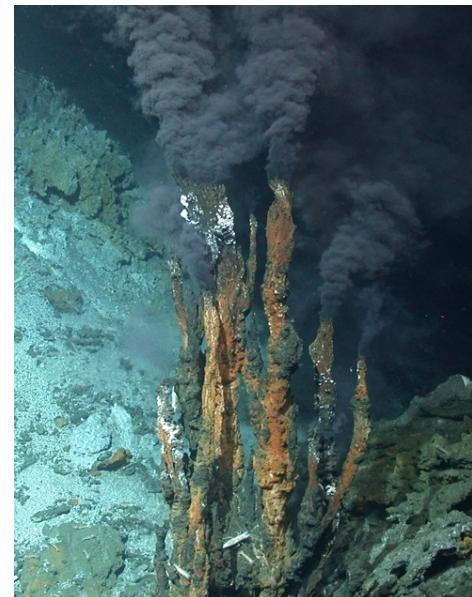


Why use numerical models?

Link observations to processes



Guo et al., 2020



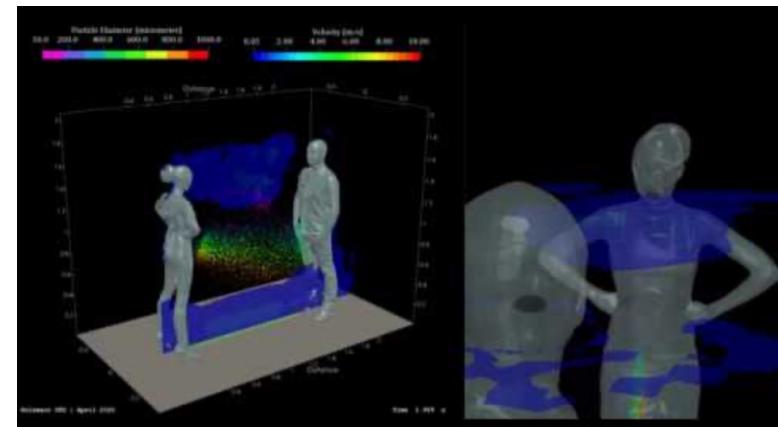
unknown source – probably Geomar



Why use OpenFOAM?

- ❖ OpenFOAM is an open-source collection of over 200 programs for multi-physics numerical simulations
- ❖ Main use is in computational fluid dynamics
- ❖ It's written in C++ and is fully parallelized
- ❖ treats 3D geometries
- ❖ easily modifiable
- ❖ has an active community

[youtube channel of Tobi Holzmann](#)



Further readings and documentations

- ❖ great lectures by Cyprien Soulaine (www.cypriensoulaine.com)
 - ❖ ...parts of this material are based on his slides
- ❖ <https://cfd.direct/openfoam/documentation/>
- ❖ Videos at <https://holzmann-cfd.com> and his youtube channel
- ❖ <https://www.cfd-online.com/Forums/>
- ❖ Code documentation at <https://cpp.openfoam.org/v8/>

OpenFOAM flavors

- ❖ OpenFOAM is not a single software but a collection of 200+ programs/tools
- ❖ OpenFOAM also comes in many different flavors
 - ❖ Foundation version (openfoam.org)
 - ❖ ESI OpenFOAM (openfoam.com)
 - ❖ foam-extend
- ❖ Downside is that some of these versions are incompatible...
- ❖ We use a precompiled docker container of the Foundation version

- ❖ We have prepared a docker container with a suitable installation
 - ❖ find it at www.hydrothermalfoam.com
-

HYDROTHERMALFOAM **QUICK START**



Pre-Processing: blockMesh, snappyHexMesh, etc.

Solution stage: simpleFoam, buoyantPimpleFoam, etc.

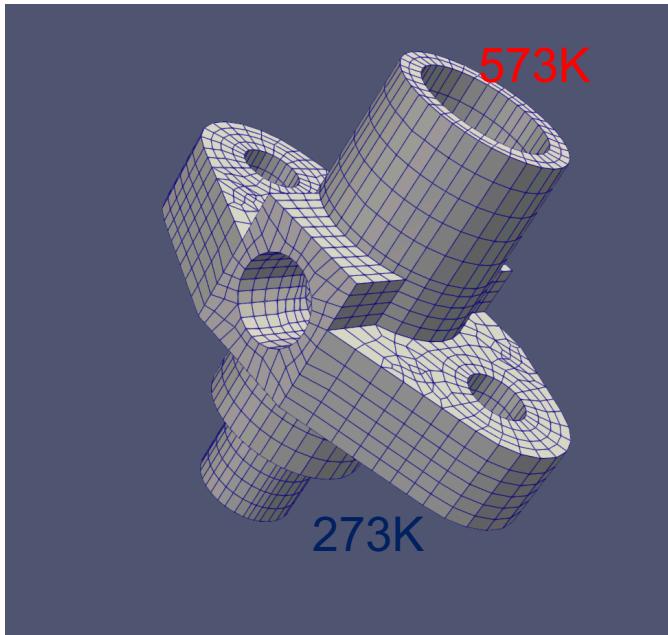
Post-Processing: paraView, python, gnuplot, etc.

OpenFOAM workflow

```
laplacianFoam — lruepke@m...  
[(base) →  laplacianFoam tree  
  
└ flange  
    └ 0  
        └ T  
    └ Allclean  
    └ Allrun  
    └ constant  
        └ transportProperties  
    └ flange.ans  
    └ system  
        └ controlDict  
        └ fvSchemes  
        └ fvSolution  
  
4 directories, 8 files  
(base) →  laplacianFoam
```

- ❖ An OpenFOAM case consists of a certain directory structure
- ❖ Most things in OpenFOAM are organized in so-called dictionaries
- ❖ A simple OpenFOAM case looks like this
 - ❖ 0 has all the information of the initial conditions
 - ❖ constant has all “constant” properties, like transport properties
 - ❖ system contains all files that control the case (solver, time stepping, output writing, etc.)

A first test case – heat diffusion



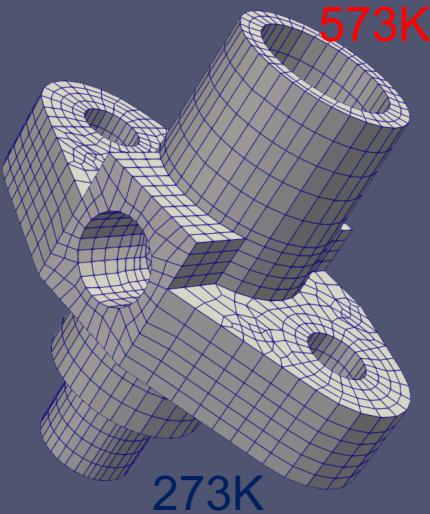
We will use one of OpenFOAM's tutorials, the heating of a flange.

It's a simple test case where a flange responds to constant temperature boundary conditions.

The equation we solve is this:

$$\frac{\partial T}{\partial t} = \nabla \cdot D \nabla T$$

A first test case – heat diffusion



What do we need?

- ❖ a mesh
- ❖ boundary/initial conditions
- ❖ a solver
- ❖ simulations controls (time step, run time, etc.)
- ❖ post-processing

A first test case – heat diffusion

Copy the OpenFOAM case into your working directory

```
cp -r $FOAM_TUTORIALS/basic/laplacianFoam/flange .
```

Convert ansys file to OpenFOAM format

```
ansysToFoam flange.ans -scale 0.001
```

Check out the mesh in paraview

```
touch a.foam; paraview a.foam
```

Check how the directory structure has changed

A first test case – heat diffusion

```
laplacianFoam — lruepke@m...  
[(base) +] laplacianFoam tree  
.  
└── flange  
    ├── 0  
    │   └── T  
    ├── Allclean  
    ├── Allrun  
    ├── constant  
    │   └── transportProperties  
    ├── flange.ans  
    └── system  
        ├── controlDict  
        ├── fvSchemes  
        └── fvSolution  
  
4 directories, 8 files  
(base) +] laplacianFoam
```

```
flange — lruepke@muhs-nb017mc  
[+] flange tree  
.  
└── flange  
    ├── 0  
    │   └── T  
    ├── Allclean  
    ├── Allrun  
    ├── a.foam  
    └── constant  
        ├── polyMesh  
        │   ├── boundary  
        │   ├── faceZones  
        │   ├── faces  
        │   ├── neighbour  
        │   ├── owner  
        │   └── points  
        └── transportProperties  
    ├── flange.ans  
    └── system  
        ├── controlDict  
        ├── fvSchemes  
        └── fvSolution  
  
4 directories, 15 files  
[+] flange
```

The polyMesh directory contains all the mesh information.

Organized as Finite Volumes

A first test case – heat diffusion

```
Users > lruepke > OpenFOAM_data > Lecture > flange > 0 > T
```

```
8 FoamFile
9 {
10     version    2.0;
11     format      ascii;
12     class       volScalarField;
13     object      T;
14 }
15 // * * * * *
16 dimensions [0 0 0 1 0 0 0];
17 internalField uniform 273;
18
19 boundaryField
20 {
21     patch1
22     {
23         type zeroGradient;
24     }
25
26     patch2
27     {
28         type fixedValue;
29         value uniform 273;
30     }
31 }
```

Boundary conditions

- ❖ boundary/initial conditions are in the 0 directory
- ❖ checkout the 0/T file
- ❖ Flange is initially at 273K
- ❖ patch2 and patch4 have the boundary conditions

Units! [Kg m s K mol A cd]

A first test case – heat diffusion

Transport properties (here diffusivity)

- ❖ check constant/transportProperties
- ❖ diffusivity has units of m²/s

```
Users > Iruepke > OpenFOAM_data > Lecture > flange > constant > transportProperties
1  *-----* C++ *-----*
2  =====
3  \\\ / Field      | OpenFOAM: The Open Source CFD Toolbox
4  \\\ / Operation   | Website: https://openfoam.org
5  \\\ / And        | Version: 7
6  \\\ \ Manipulation |
7  *-----*
8 FoamFile
9 {
10    version    2.0;
11    format     ascii;
12    class      dictionary;
13    location   "constant";
14    object     transportProperties;
15 }
16 // ****
17 DT [0 2 -1 0 0 0] 4e-05;
18 // ****
19
20
21 // ****
22
```

Units! [Kg m s K mol A cd]

A first test case – heat diffusion

```
Users > lruepke > OpenFOAM_data > Lecture > flange >
1  /*
2   *----- C++ -----
3   \\\ / F ield      | OpenFOAM: T
4   \\\ / O peration  | Website: h
5   \\\ / A nd        | Version: 7
6   \\\ M anipulation |
7  */
8 FoamFile
9 {
10    version    2.0;
11    format     ascii;
12    class      dictionary;
13    location   "system";
14    object     controlDict;
15 }
16 // * * * * *
17 application laplacianFoam;
18
19 startFrom latestTime;
20
21 startTime 0;
22
23 stopAt endTime;
24
25 endTime 3;
26
27 deltaT 0.005;
28
29 writeControl runTime;
30
31 writeInterval 0.1;
32
33 purgeWrite 0;
34
35 writeFormat ascii;
```

system/controlDict contains all the simulation controls like solver, start/end times, time step, output writing, etc.

Check also system/fvSolution and system/fvSchemes for extra options

But which equation are we solving, what is the solver really doing?

Check cpp.openfoam.org for laplacianFoam
this is the build-in solver we use!

A first test case – heat diffusion

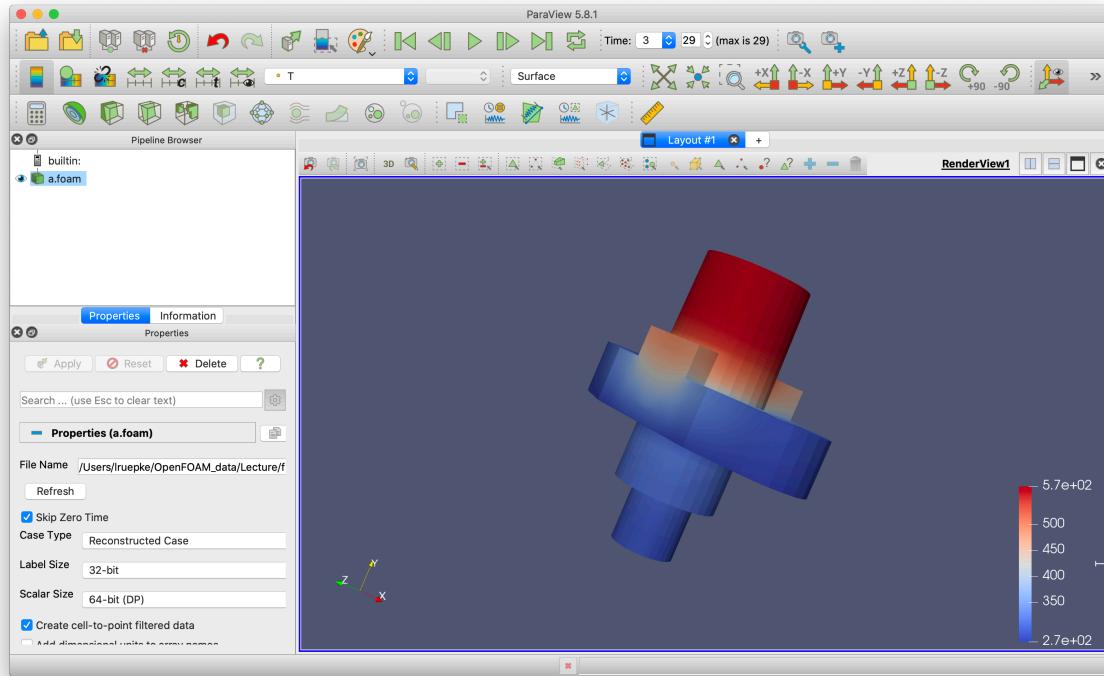
```
// **** *  
int main(int argc, char *argv[]){  
    #include "setRootCaseLists.H"  
    #include "createTime.H"  
    #include "createMesh.H"  
    simpleControl simple(mesh);  
    #include "createFields.H"  
// **** *  
Info<< "\nCalculating temperature distribution\n" << endl;  
while (simple.loop(runTime))  
{  
    Info<< "time = " << runTime.timeName() << nl << endl;  
    while (simple.correctNonOrthogonal())  
    {  
        fvScalarMatrix TEqn  
        (  
            fvm::ddt(T) - fvm::laplacian(DT, T)  
            ==  
            fvOptions(T)  
        );  
        fvOptions.constrain(TEqn);  
        TEqn.solve();  
        fvOptions.correct(T);  
    }  
    #include "write.H"  
    Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s"  
    << " ClockTime = " << runTime.elapsedClockTime() << " s"  
    << nl << endl;  
}  
Info<< "End\n" << endl;
```

OpenFOAM magic

Checkout how simple the implementation of the Laplace equation is!

$$\frac{\partial T}{\partial t} = \nabla \cdot D T \nabla T$$

A first test case – heat diffusion



Post-processing

Look at the solution with Paraview

2D HydrothermalFoam Simulation

Preparations:

copy base case from tutorial directory (in docker)

```
→ 2d pwd  
/home/openfoam/hydrothermalfoam-master/cookbooks/2d  
→ 2d ls  
Regular2DBox gmesh2DBox  
→ 2d cp -r Regular2DBox $HOME/HydrothermalFoam runs
```

```
→ Regular2DBox tree
.
|-- 0 → Initial condition
|   |-- T
|   |-- p
|   '-- permeability
|-- clean.sh → Clean case
|-- constant
|   |-- g
|   '-- thermophysicalProperties
|-- run.sh → Run case
|-- system
|   |-- blockMeshDict
|   |-- controlDict
|   |-- fvSchemes
|   '-- fvSolution
.
→ Geometry & mesh define

3 directories, 11 files
→ Regular2DBox
```

1. Geometry and mesh definition in `blockMeshDict`

```
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       blockMeshDict;
}

// ****
convertToMeters 1;
Lx 2000;      //variable definition
ymin -3000;
ymax -2000;
Lz 1;
vertices      //vertices definition
(
    (0      $ymin  0)  //coordinate of vertex 0
    ($Lx   $ymin  0)  //coordinate of vertex 1
    ($Lx   $ymax   0) //coordinate of vertex 2
    (0      $ymax   0) //coordinate of vertex 3
    (0      $ymin   $Lz)//coordinate of vertex 4
    ($Lx   $ymin   $Lz)//coordinate of vertex 5
    ($Lx   $ymax   $Lz)//coordinate of vertex 6
)
```

Head of **Dict** file, keywords **class**, **object**

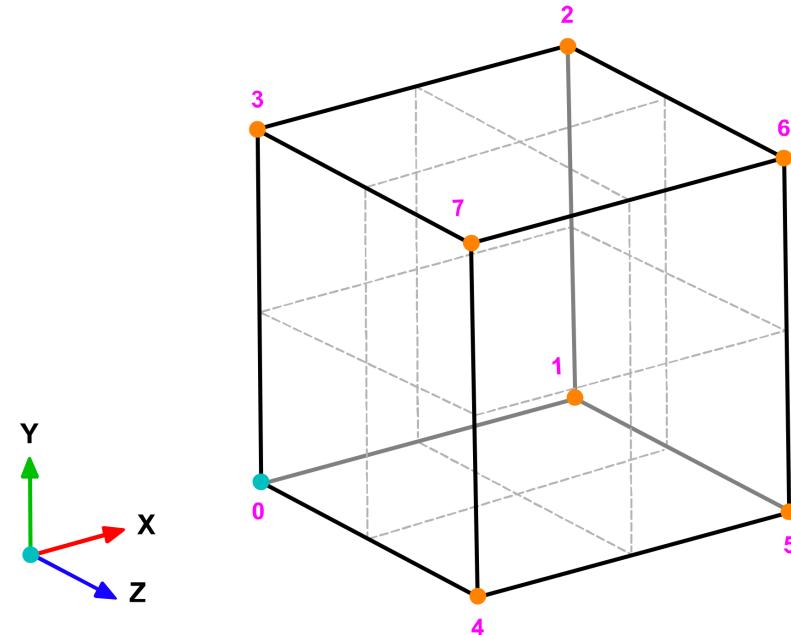
Useful variables definition

- Syntax (similar to C++)
`varName varValue;`
- Macro support
`var3 #calc "$var1 + $var2";`

1. Geometry and mesh definition in `blockMeshDict`

Vertices coordinates definition

```
32 vertices
33 (
34     ($xmin $ymin $zmin) 0
35     ($xmax $ymin $zmin) 1
36     ($xmax $ymax $zmin) 2
37     ($xmin $ymax $zmin) 3
38     ($xmin $ymin $zmax) 4
39     ($xmax $ymin $zmax) 5
40     ($xmax $ymax $zmax) 6
41     ($xmin $ymax $zmax) 7
42 );
```



1. Geometry and mesh definition in `blockMeshDict`

Blocks definition

Vertices order

```
44 blocks
45 (
46     hex (0 1 2 3 4 5 6 7) ($xcells $ycells $zcells) simpleGrading (1 1 1)
47 );
```

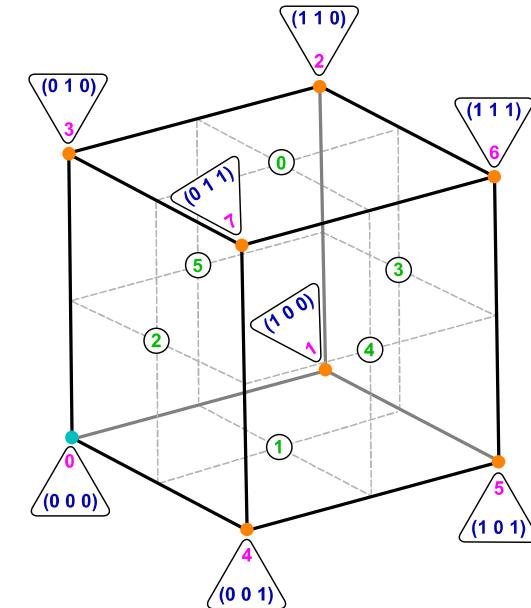
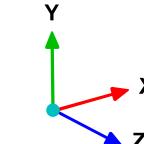
Cell number
in each direction

Expansion ration
in each direction

Vertices order is
Important !!!

| VERTICES | |
|----------|-------|
| 0 | 0 0 0 |
| 1 | 1 1 0 |
| 2 | 1 1 0 |
| 3 | 0 1 0 |
| 4 | 0 0 1 |
| 5 | 1 0 1 |
| 6 | 1 1 1 |
| 7 | 0 1 1 |

| BLOCK (HEX) | |
|-----------------|---------|
| 0 1 2 3 4 5 6 7 | |
| FACES | |
| 0 | 3 7 6 2 |
| 1 | 1 5 4 0 |
| 2 | 0 4 7 3 |
| 3 | 2 6 5 1 |
| 4 | 0 3 2 1 |
| 5 | 4 5 6 7 |



1. Geometry and mesh definition in `blockMeshDict`

Boundary patch definition

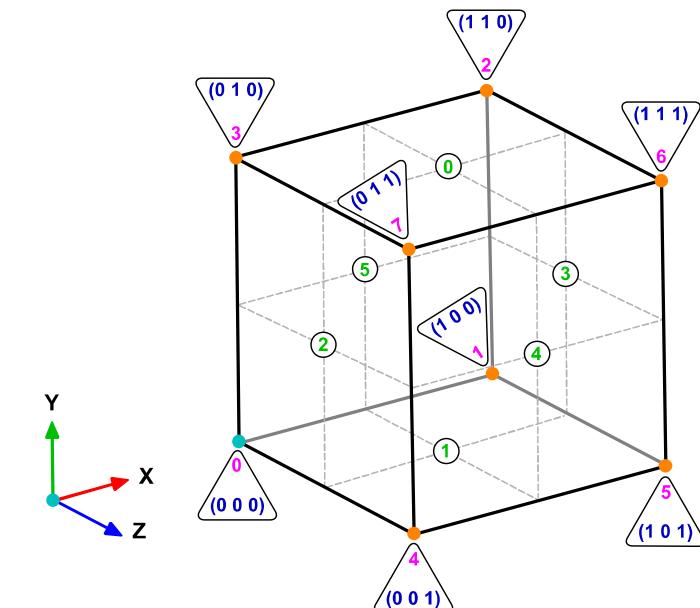
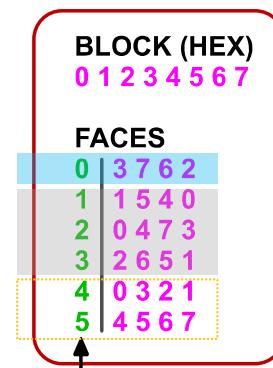
```

boundary
(
    left      //patch name
    {
        type patch;
        faces   //face list
        (
            (0 4 7 3)
        );
    }
    right
    {
        type patch;
        faces
        (
            (2 6 5 1)
        );
    }
    top
    {
        type patch;
        faces
        (
            (3 7 6 2)
        );
    }
    bottom
    {
    }
);
  
```

Type

Name

Connectivity

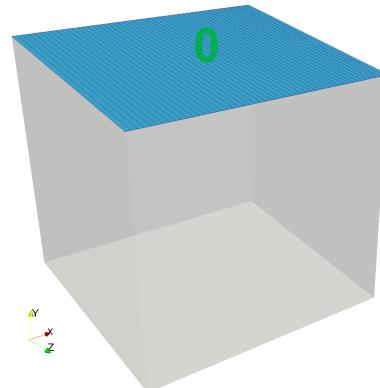


1. Geometry and mesh definition in `blockMeshDict`

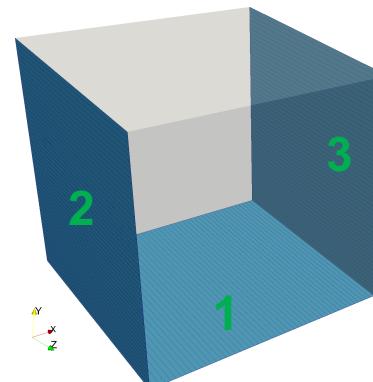
Boundary patch definition



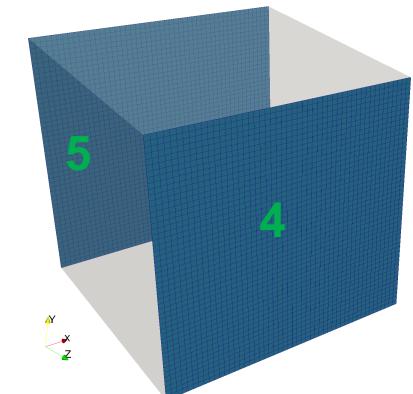
top



left right bottom



frontAndBack



```
frontAndBack //patch name
{
    type empty;
    faces //face list
    (
        (0 3 2 1) //back face
        (4 5 6 7) //front face
    );
};
```

2D HydrothermalFoam Simulation

3. Initial condition and boundary condition – θ/p

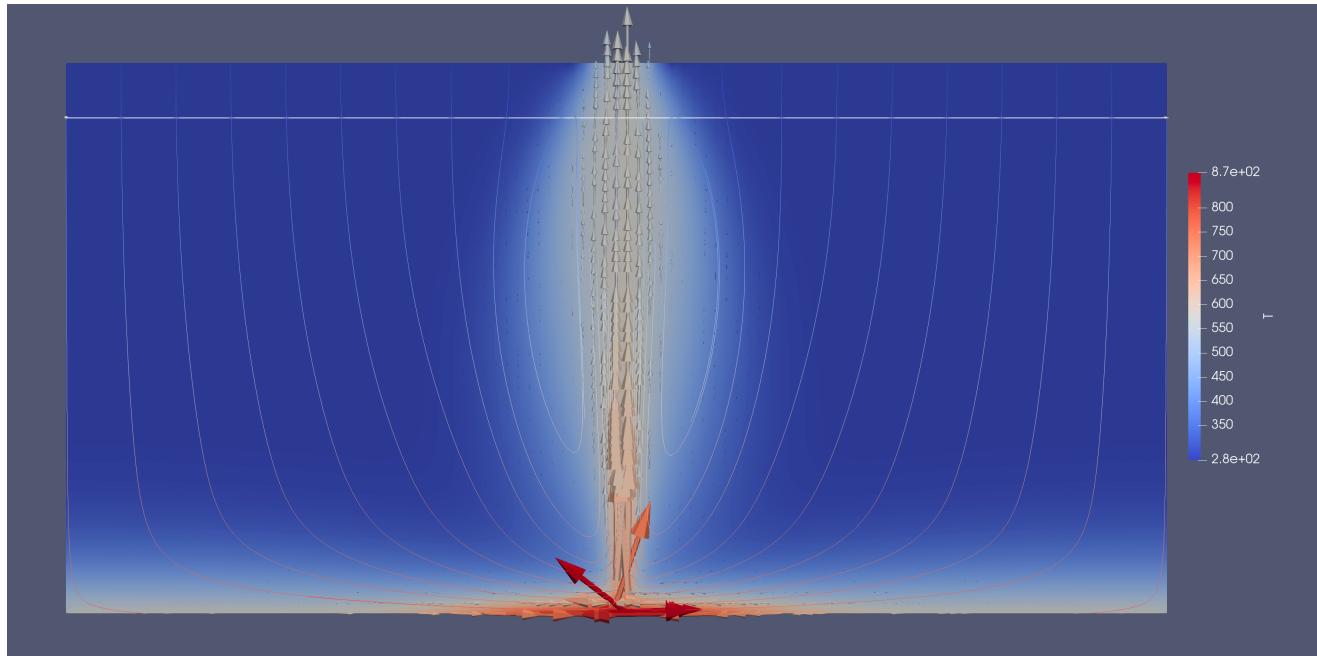
```
dimensions      [1 -1 -2 0 0 0];  
  
internalField  uniform 300e5;  
  
boundaryField  
{  
    left  
    {  
        type      noFlux;  
    }  
    right  
    {  
        type      noFlux;  
    }  
    top  
    {  
        type      fixedValue;  
        value    uniform 300e5;  
    }  
    bottom  
    {  
        type      noFlux;  
    }  
    frontAndBack  
    {  
        type      empty;  
    }  
}
```

Dimensions of the field $Kg\ m^{-1}\ s^{-2} = Pa$

Uniform initial conditions.

noFlux means that pressure gradient is such that there is no flow through these boundaries-

3. Results



2D Fault-controlled flow

[zguoch / HydrothermalFoam_iMOVE](https://github.com/zguoch/HydrothermalFoam_iMOVE)

Code Issues Pull requests Actions Projects Wiki Security Insights

master 1 branch 0 tags Go to file Add file Code

zguoch add readme to each case 61d9454 16 hours ago 7 commits

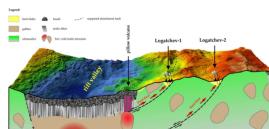
- Logatchev_DF10 add readme to each case 16 hours ago
- Logatchev_DF150 add readme to each case 16 hours ago
- Logatchev_DF50 add readme to each case 16 hours ago
- images add readme to each case 16 hours ago
- readme.md add readme to each case 16 hours ago

readme.md

Case study using HydrothermalFoam

Here we share a case study of Logatchev-1 hydrothermal system.

Geological background of Logatchev-1 field



Geophysical data of Logatchev-1 field

About

case files, materials for iMOVE hands-on work shop

Readme

Releases

No releases published Create a new release

Packages

No packages published Publish your first package

Languages

| | |
|-------------|------------|
| C++ 65.3% | GLSL 19.8% |
| Shell 14.9% | |

TERMINAL PROBLEMS OUTPUT DEBUG CONSOLE

```
→ HydrothermalFoam_runs git clone https://github.com/zguoch/HydrothermalFoam_iMOVE.git
```