Geoscientific
Model Development
Discussions

Open Access

EGU

Interactive comment on "*HydrothermalFoam* v1.0: a 3-D hydro-thermo-transport model for natural submarine hydrothermal systems" *by* Zhikui Guo et al.

Zhikui Guo et al.

zguo@geomar.de

1 Technical corrections

We have discovered a number of technical issues in the original version of HydrothermalFoam v1.0.0. For the convenience of the reviewers and early users of *HydrothermalFoam*, we have fixed those issues and document the changes in this "short comment". In particular, we have fixed a few issues in the benchmark and cookbook cases, both in the docker container and in the gitlab source directory, and have have updated the *thermoPhysical model*. The details are described below.

C1

1.1 Technical issues of HydrothermalFoam v1.0.0

We have discovered a few technical issues in the first version of *HydrothermalFoam* which is linked in the discussion paper.

- Mesh generation using gmsh

  The original run.sh of the 2D benchmark example was missing a command to generate the mesh using gmsh. As a consequence, the run.sh script returned an error. This issue was fixed by adding the respective gmsh command (gmsh gmsh/box.geo -3 -o gmsh/box.msh -format msh22) to run.sh, see the new version of run.sh.

- Bash syntax compatibility issues

  Some of the run.sh scripts in the cookbook and benchmark directories contain a command (. clean.sh) to clean the tutorial case, e.g. cookbooks/2d/Regular2DBox. This command works under our Mac OS test environment but not in the docker container or other linux systems. Therefore, this command was changed to ./clean.sh.

- 0/permeability file missing

  The permeability field input file is missing in the **0** directory in some cases, e.g. benchmarks/HydrothermalFoam/2d. This is due to 0/permeability being incorrectly written into the .gitignore file in the case directory. This has been fixed in the latest version of HydrothermalFoam.

1.2 Updates to the thermoPhysical model

The thermoPhysical model implements the pure water equation-of-state and is also the natural place where all other material properties, i.e. the matrix properties of the

porous media, should be implemented. However, the original version v1.0.0 used the constant/transportProperties file to implemented the matrix properties (*porosity, kr, cp_rock, rho_rock*). In this update, we have implemented all porous media properties into the thermoPhysical model. Therefore, in v1.0.1 or the latest version, the transportantProperties file is no longer needed by the *HydrothermalSinglePhaseDarcy* solver. The porous medium properties are inputed as a sub-dictionary (named porousMedia) in constant/thermophysicalProperties. The complete thermophysicalProperties file of the latest version is listed below,

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      thermophysicalProperties;
}
thermoType
{
    type            htHydroThermo;
    mixture         pureMixture;
    transport       IAPWS;
    thermo          IAPWS;
    equationOfState IAPWS; //Boussinesq
    specie          specie;
    energy          temperature;
}

mixture
```

```
{
    specie
    {
        molWeight       18;
    }
    porousMedia
    {
        cp_rock cp_rock [0 2 -2 -1 0 0 0] 880;
        rho_rock rho_rock [1 -3 0 0 0 0 0] 2700;
        porosity porosity [0 0 0 0 0 0 0] 0.1;
        kr kr [1 1 -3 -1 0 0 0] 2;
    }
}
```

See also the latest version of manual.

1.3 Change log of HydrothermalFoam v1.0.1

1. Fixed few issues in benchmarks and cookbooks.

2. Update thermoPhysical model and deprecated transportantProperties file for *HydrothermalSinglePhaseDarcy* solver.

3. Add serial and parallel computing performance example into cookbooks, see the latest manual for the comparison results.

4. Update docker image with the latest source code.

2 How to update HydrothermalFoam in the docker container

The docker image of *HydrothermalFoam* has been updated on **July 19, 2020** and pushed to the Docker Hub repository. Note that it will not be updated frequently in the future, but the *HydrothermalFoam* will be.

There are two ways to update *HydrothermalFoam* source code. The first one is using git to clone and pull the latest version of the *HydrothermalFoam* source code. The second one is using the latest version of the docker image and run the bash script of getHydrothermalFoam_latest.sh in the home directory of the docker container.

We recommend the second approach for users who are not familiar with git. **If you are installing docker image of HydrothermalFoam for the first time**, please just follow the installing instructions in the manual or watch the tutorial video for the beginners to install the docker image and start a docker container. **If you have already installed docker image of HydrothermalFoam before July 19, 2020**, please follow the steps below to use the latest version of HydrothermalFoam.

Open terminal (if Mac OS or Linux) or Windows PowerShell and run commands (text in orange color) as the following steps.

1. Pull the latest version of docker image of *HydrothermalFoam*: docker pull zguo/hydrothermalfoam

2. Check for existing docker containers: docker containers ls –all

3. Remove the **hydrothermalfoam** container if it exist: docker rm hydrothermalfoam

4. Reinstall docker container by running commands in installMacHydrothermalFoam.sh for Mac OS and Linux, or running installWindowsHydrothermalFoam.ps1 for Windows system. See also **installation** section in the manual.

5. Start the docker container of *hydrothermalfoam*: docker start hydrothermalfoam

6. Attach the container of *hydrothermalfoam*: docker attach hydrothermalfoam

7. Getting the latest version of *HydrothermalFoam*: ./getHydrothermalFoam_latest.sh. Note that the script will download and compile the latest source code automatically. The folder name of the latest source code is **HydrothermalFoam-master** and the previous version existed in the docker container will be renamed as **HydrothermalFoam-master_backup** which can be deleted by user if there is not important user data in the folder.

3 Bug report

We keep working on developing and improving the source code and will continue to find bugs. And so will the users of HydrothermalFoam. We would like to encourage all users to submit bug reports using the issue tracker (https://gitlab.com/gmdpapers/hydrothermalfoam/-/issues) of the GitLab repository of HydrothermalFoam.