Thermo-Fluid-Structure Interaction Based on the Fictitious Domain Method: Application to Dry Rock Simulations

Nasibeh Hassanjani-khoshkroud¹,⁴, Maria G. C. Nestola¹,², Patrick Zulian¹, Cyrill Von Planta¹, Daniel Vogler³, Harald Köstler¹, Rolf Krause¹

¹Institute of Computational Science, USI, Via Giuseppe Buffi 13, 6900, Lugano Switzerland
²Institute of Geochemistry and Petrology, ETH Zurich, Clausiusstr. 25, 8092 Zurich, Switzerland
³Institute of Geophysics, ETH Zurich, Sonneggstr. 5, 8092 Zurich, Switzerland
⁴Institute of System Simulation, FAU, Cauerstraße 11, 91058, Erlangen Germany

Keywords: Thermal-fluid-structure interaction, enhanced geothermal systems, fictitious domain method, variational transfer.

ABSTRACT

Enhanced Geothermal Systems (EGS) generate geothermal energy without the need for natural convective hydrothermal resources by enhancing permeability through hydraulic fracturing. EGS involve complex and highly nonlinear multiphysics processes and face several technical challenges governing productivity and associated risks in a wide range of reservoir engineering problems. Numerical simulations offer a unique opportunity to improve the hydraulic stimulation design and investigate their long-term performance. In this work, we present a thermo-hydro-mechanical coupling based on the fictitious domain method. Fictitious domain methods include nonconforming mesh approaches that rely on the use of different discretizations for the solid and the fluid domain. We consider the governing equations of linear thermo-elasticity to model the heat transfer in dry and hot rocks. Navier-Stokes equations and heat convection equations are adopted for describing the thermal flux in an incompressible fluid flow. The two problems are coupled in a staggered manner through the variational transfer. Indeed, the use of the variational transfer allows simplifying the setup of fracture simulations with complex surfaces embedded in the fluid domain, thus avoiding the inconvenient mesh generation required by the boundary fitted methods. The presented computational framework is first verified with analytical two-dimensional benchmarks, and then employed to simulate the thermo-hydro-mechanical coupling between fluid and idealized fracture asperities.

1. INTRODUCTION

Thermal-Fluid-Structure Interaction (TFSI) is an essential process in enhanced geothermal systems (EGS) and also in the engineering of them (Li et al. 2016; Cacace and Jacquey 2017; Li et al. 2019; Lima et al. 2019). Thus, numerical simulations may play a crucial role in understanding fluid flow through fractured rocks under thermo-hydro-mechanical conditions. In this regard, numerical models of fractured reservoirs have become an increasingly active area of research (Yin and Liu 2015; Li et al. 2016; Cacace and Jacquey 2017; Li et al. 2019) mainly due to the computational and numerical challenges arising from the complex and nonlinear multiphysics processes occurring in EGS (Taron and Elsworth 2009, Vogler et al. 2018).

In the literature, several methods have been introduced, which can be classified in conforming and non-conforming approaches. The complexity of conforming approaches is related to the mesh generation, whereas non-conforming mesh methods require the use of specific numerical techniques such as mortar methods, extended finite element discretizations, and PDE constrained optimization.

The approach presented in this contribution is inspired by the fictitious domain (FD) method, which has been introduced in (Glowinski et al. 1999) to simulate the movement of particles into a fluid and then extended to elastic bodies in (Yu et al. 2005). The main advantage of the FD approach is that it allows for the discretization of the fluid problem on a fixed grid by adopting an Eulerian description, whereas the solid structure is tracked in a Lagrangian fashion. The coupling conditions between the two physics is prescribed on the fluid-solid interface (Glowinski et al. 1999) or over the entire overlapping region (Boffi et al. 2003; Hesch et al. 2014) by using Lagrange multipliers or penalization techniques.

In this paper, we present a Thermo-Fluid-Structure interaction approach, which relies on the fluid-structure interaction (FSI) formulation introduced in (Nestola et al. 2016) to model hyperelastic structures embedded in complex flows, and recently employed in (Planta et al. 2019) for the simulation of fluid flow in fracture asperities. These FSI methodologies are further extended to model the heat conduction in the solid, the heat convection in the fluid flow, and the heat exchange between the two physics.

In particular, we employ a staggered strategy where all physical quantities of interest are transferred between the fluid and the solid mesh through the fully parallel variational approach proposed in (Zulian et al. 2016). We employ a penalization technique to couple the Navier-Stokes equations with the constraints prescribing continuity of temperature and velocity. Moreover, the solid mechanics equations are slightly modified to include the force exerted by the fluid to the solid, and the heat exchanged between the two physics.
Hassanjakshkroud et al.

The proposed methodology is verified with analytical benchmarks and finally employed to simulate fluid flow and heat transfer in simplified fracture asperities.

2. THERMO-FLUID-STRUCTURE-INTERACTION

The key ingredients of the proposed TFSI framework are 1) the thermo-elasticity problem, 2) the thermo-fluid dynamics equations, and 3) the coupling of the two subproblems.

2.1 Thermo-elasticity problem

Thermo-elasticity is the study of the elastic behavior of a body subject to temperature changes and prescribed external forces. In particular, we restrict our attention to linear elastic materials, such as fractured rocks, whose deformations are linearly proportional to the magnitude of applied stress.

We define \( \Omega_s \subset \mathbb{R}^n, n \in \{2,3\} \) as a bounded domain representing the region occupied by the solid body whose boundary \( \partial \Omega_s \) is split into the Dirichlet \( \partial \Omega_{Ds} \) and Neumann \( \partial \Omega_{Ns} \) parts such that \( \partial \Omega_{Ds} \cup \partial \Omega_{Ns} = \partial \Omega_s \) and \( \partial \Omega_{Ds} \cap \partial \Omega_{Ns} = \emptyset \) (as shown in Figure 1). The displacement field \( d_s \) and the temperature field \( T_s \) of a solid structure are obtained by solving the following equations (Nowinski 1978; Danowski et al. 2013):

\[
\begin{align*}
\rho_s \ddot{d}_s - \nabla \cdot \sigma_s &= \rho_s F_s & \text{in } \Omega_s \\
\rho_s c_s \dot{T}_s + \nabla \cdot (-k_s \nabla T_s) - T_{\text{ref}} B_l: \dot{\varepsilon}_m &= 0 & \text{in } \Omega_s \\
\sigma_s &= 2\mu \varepsilon_m + \lambda \varepsilon_m I + \beta l (T_s - T^*_s) & \text{in } \Omega_s \\
d_s &= d_{D_s}, & \text{on } \partial \Omega_{Ds} \\
\sigma_s n_s &= \bar{T}_{Ns}, & \text{on } \partial \Omega_{Ns} \\
d_s = d_{\text{as}}, & u_s = d_{\text{as}}, & T_s = T_{\text{as}}, & t = 0 \forall x \in \Omega_s
\end{align*}
\]

where \( \rho_s \) is the density, \( u_s = \dot{d}_s \) is the solid velocity vector field, \( \sigma_s \) is the solid Cauchy stress tensor, \( \varepsilon_m \) is the linear strain tensor, \( F_s \) is a volumetric density force, \( c_s \) is the specific heat capacity, \( \beta \) is the thermal expansion coefficient, \( \alpha = -(2\mu + 3\lambda) \) is the stress-temperature modulus, \( T^*_s \) is the prescribed stress-free temperature, \( I \) is the second rank identity tensor, \( k_s \) is the heat conductivity (here assumed homogeneous and isotropic), \( \beta = \alpha T^*_s \) is the thermal expansion coefficient, \( \varepsilon_m \) is the linear strain tensor, \( T_s \) is the temperature field, \( T^*_s \) is the prescribed stress-free temperature, \( I \) is the second rank identity tensor, \( k_s \) is the heat conductivity (here assumed homogeneous and isotropic), \( \beta = \alpha T^*_s \) is the thermal expansion coefficient, \( \sigma_s \) is the Cauchy stress tensor, \( \varepsilon_m \) is the linear strain tensor, \( F_s \) is a volumetric density force, \( \rho_s \) is the density, \( \mu_f \) is the fluid viscosity, \( k_f \) is the fluid heat conductivity (here assumed homogeneous and isotropic), \( c_f \) is the fluid heat capacity, \( I \) is the identity tensor, \( t \) is the

2.2 Thermo-fluid dynamics problem

We introduce the Navier-Stokes (NS) equations defined in a Eulerian framework and couple them with the energy equation to model heat transfer within a fluid. We restrict our attention to incompressible Newtonian flows.

Let \( \Omega_f \subset \mathbb{R}^n, n \in \{2,3\} \) be a bounded fluid domain whose boundary \( \partial \Omega_s \) is split into the Dirichlet \( \partial \Omega_{DF} \) and Neumann \( \partial \Omega_{NF} \) parts such that \( \partial \Omega_{DF} \cup \partial \Omega_{NF} = \partial \Omega_f \) and \( \partial \Omega_{DF} \cap \partial \Omega_{NF} = \emptyset \) (as shown in Figure 2). The pressure field \( p_f \), the velocity field \( u_f \), and the temperature field \( T_f \) are obtained by solving the following equations (Anderson and Wendt 1995):

\[
\begin{align*}
\nabla \cdot u_f &= 0 & \text{in } \Omega_f \\
\rho_f u_f + p_f (u_f \cdot \nabla u_f) - \nabla \cdot \sigma_f &= \rho_f F_f & \text{in } \Omega_f \\
\rho_f c_f \dot{T_f} + u_f \cdot \nabla T_f + \nabla \cdot (-k_f \nabla T_f) - \sigma_f : \nabla u_f &= 0 & \text{in } \Omega_f \\
\sigma_f &= -p_f I + \mu_f (\nabla u_f + \nabla u_f^T) & \text{in } \Omega_f \\
u_f &= u_{\text{DF}}, & T_f = T_{\text{DF}}, & t = 0 \forall x \in \Omega_f
\end{align*}
\]

where \( \sigma_f \) is the Cauchy stress for Newtonian fluids, \( F_f \) is a volumetric density force, \( \rho_f \) is the fluid density, \( \mu_f \) is the fluid viscosity, \( k_f \) is the fluid heat conductivity (here assumed homogeneous and isotropic), \( c_f \) is the fluid heat capacity, \( I \) is the identity tensor, \( t \) is the
time and \( n_f \) is the outward unit normal to the fluid boundary. The superscript \( (\cdot) \) refers to the first time-derivative. The velocity \( u_{\text{df}} \) and the temperature \( T_{\text{df}} \) are the prescribed values at the Dirichlet boundary. The stress \( \tilde{s}_f \) and the heat flux \( q_{\text{df}} \) are prescribed values at the Neumann boundary. The initial values of the velocity vector field and temperature field are defined by \( u_{\text{df}} \) and \( T_{\text{df}} \), respectively.

![Figure 1: Solid domain.](image1)

![Figure 2: Fluid domain.](image2)

2.3 Coupling conditions

Similar to former FSI approaches based on the fictitious domain (Hesch et al. 2014; Nestola et al. 2019; Planta et al. 2019), the solid and fluid dynamics problems are solved on different domains, and then coupled through proper constraints imposed on the overlapping region, i.e. the region shared by the fluid and the solid. This is done in a staggered manner by means of a fixed-point iteration scheme to satisfy the coupling conditions. The fluid problem is formulated in Eulerian coordinates, while the solid problem is formulated in Lagrangian coordinates. We employ a penalization technique to couple the Navier-Stokes equations with the constraints prescribing continuity of temperature and velocity over the entire overlapping region (Planta et al. 2019). Then, we modify the solid mechanics equations to include the forces exerted by the fluid to the solid to reach the mechanic equilibrium, and the heat transferred from the fluid to the solid to reach the thermodynamic equilibrium. According to (Hesch et al. 2014) the inertial terms of the solid problem are evaluated by using the difference between the solid and the fluid density. The internal stresses and heat conduction of the fluid should also be subtracted from the solid contributions. However, this operation can be neglected in most applications since internal fluid stresses and fluid conductivity are considerably smaller than their solid counterparts.

2.3.1 Strong formulation

We embed the current configuration of the solid domain \( \Omega_s(t) \) into the fluid domain \( \Omega_f \) such that \( \partial \Omega_f \cap \partial \Omega_s(t) = \emptyset \) (Figure 3), and denote the overlapping region as \( \Omega_{fsi}(t) = \Omega_f \cap \Omega_s(t) \). The coupling TFSI conditions read as follows

![Figure 3: TFSI scenario with a solid domain \( \Omega_s(t) \) embedded in a fluid domain \( \Omega_f \)](image3)

Here \( F_{f \rightarrow s} \) and \( F_{s \rightarrow f} \) are the forces exerted by the fluid to the solid and from the solid to the fluid, respectively, whereas \( Q_{f \rightarrow s} \) and \( Q_{s \rightarrow f} \) is the heat exchanged from the fluid to the solid, and vice versa. Equations (14) and (15) ensure the mechanic and the thermodynamic equilibrium, respectively.

2.3.2 Weak formulation

Let us introduce the following function spaces:

\[ S^d(\Omega_f) = \{ v_m \in V_{Df}, v_m = 0 \text{ on } \partial \Omega_{Df} \} \]

\[ S^T(\Omega_f) = \{ v_T \in V_{Df}, v_T = 0 \text{ on } \partial \Omega_{Df} \} \]

\[ S^p(\Omega_f) = \{ w_p \in W_{Df}, w_p = 0 \text{ on } \partial \Omega_{Df} \} \]

\[ S^u(\Omega_f) = \{ w_u \in W_{Df}, w_u = 0 \text{ on } \partial \Omega_{Df} \} \]

Then, the weak formulation of the TFSI problem reads as follows:

\[
\int_{\Omega_s(t)} (\rho_s - \rho_f) d_s \cdot v_m dV + \int_{\Omega_s(t)} \sigma_s \cdot \nabla v_m dV - \int_{\Omega_s(t)} \tilde{F}_s \cdot v_m dA - \int_{\Omega_s(t)} \rho_s F_s \cdot v_m dV - \int_{\Omega_s(t)} F_{f \rightarrow s} \cdot v_m dV = 0 \] (18)
Hassanjanikhoshkroud et al.

\[ \int_{\Omega_f} (\rho_f c_f T_f^c \nabla T_f^c) \, dv - \int_{\Omega_f} T_{ref} \beta l : k_m v_T + \int_{\Omega_s} k_s \nabla T_s \cdot \nabla T_s \, dv - \int_{\partial \Omega_s} q_{ns} v_T \, dA - \int_{\partial \Omega_B} Q_{f-\omega} \cdot v_T \, dV = 0 \]  \hspace{1cm} (19)

\[ \int_{\Omega_f} (\nabla \cdot u_f) w_d \, dv = 0 \]  \hspace{1cm} (20)

\[ \int_{\Omega_f} \rho_f (\dot{u}_f + p_f (u_f \cdot \nabla u_f)) \cdot w_d \, dv - \int_{\partial_\Gamma_f} p_f v \cdot w_a \, dV + \int_{\partial_\Gamma_f} \mu_f \nabla u_f \cdot \nabla w_a \, dV - \int_{\partial_\Omega_s} F_f \cdot w_a \, dA - \int_{\partial \Omega_B} \bar{s}_{nf} \cdot w_a \, dA + \int_{\partial_\Omega_B} \epsilon_v^3 (u_f - u_s) w_a \, dV = 0 \]  \hspace{1cm} (21)

\[ \int_{\Omega_f} \rho_f c_f (T_f^c + u_f \cdot \nabla T_f^c) w_T \, dv + \int_{\Omega_f} k_f \nabla T_f \cdot \nabla w_T \, dv - \int_{\partial_\Omega_s} \Phi w_T \, dA - \int_{\partial \Omega_B} q_{ns} w_T \, dA + \int_{\partial_\Omega_B} \epsilon_T (T_f - T_s) w_T \, dV = 0 \]  \hspace{1cm} (22)

where \( \Phi = \sigma_f \nabla u_f \) is the viscous dissipation in the fluid flow. As already mentioned, a penalty formulation is adopted to enforce the constraints (12) and (13) with \( \epsilon \) being the penalty parameter.

### 2.3.3 Space and time discretization of the TFSI problem

For both the solid and the fluid problems we use the finite-element method for the spatial discretization. We use the Newmark method for the time discretization of the linear momentum equation of the solid problem and the implicit-Euler method for the time discretization of the remaining equations.

#### 2.3.4 Variational transfer

As previously mentioned, we solve for the thermo-elasticity and the thermo-fluid dynamics problems separately on each domain. Consequently, discrete fields need to be transferred between non-matching meshes/discretizations. This is achieved by using the parallel variational transfer algorithms presented in (Krause and Zulian 2016). Let us define a function space \( V \), with basis-functions \( v \), over the solid domain \( \Omega_s(t) \) and a function space \( W \), with basis-functions \( w \), over the fluid domain \( \Omega_f \) such that \( V \subset W \) and \( L^2 \) scalar product \( (v, w)_{L^2} = \int_v v w \, dw \) is well defined. Let \( \Pi : V \rightarrow W \), be the \( L^2 \)-projection, such that the weak-equality condition \( \int_{\Omega_f} (\Pi(v) - v) u \, dw = 0 \) holds for all \( v \in V, u \in M \), where \( M \) is a suitable Lagrange multiplier space. The discrete algebraic counterpart of the \( L^2 \)-projection \( \Pi \) is encapsulated by the matrix \( P \).

#### 2.4 Algorithm

The coupled TFSI problem is solved with a fixed-point iteration algorithm at each time step. For a more concise exposition we drop all the indices connected to the time discretization and the fixed-point iteration, except where it is necessary.

Algorithm: Fixed-point iteration

Initialization: \( l = 1, d = d_0, T_s = T_{s0}, u = u_0, p = p_0, T_f = T_{f0} \)

for \( l = 1, 2, \ldots, L_{FP} \)

\[ \text{Transfer } F_f \rightarrow s \text{ and } Q_f \rightarrow s \text{ from the fluid to the solid domain by setting: } z_f^{l \rightarrow s} := p^T z_{f \rightarrow s}, := \]  Solve solid problem.

\[ \text{Transfer } u_s, T_s \text{ to the fluid domain by setting: } (y)_s := P(y) \]  Solve fluid problem,

\[ \text{if } |x^{l+1} - x^l|_{\mathbb{R}^2} \leq \text{tol}_{FP} \]  end for

Algorithm provides an overview of the staggered approach where \( y_s \) is the solution field of the solid problem, and \( z_{f \rightarrow s} \) refers to the reaction force exerted by the fluid to the solid, and the heat exchanged between the two physics. The superscript \(^*\) refers to the projected values, whereas \( L_{FP} \) denotes the maximum number of iterations.
2.5 Implementation
Numerical simulations are carried out by using the open-source software library Utopia (Zulian et al. 2016) and MOOSE (Multiphysics Object Oriented Simulation Environment) framework (Gaston et al. 2009). The variational transfer is implemented in Utopia which makes use of LibMesh (Kirk et al. 2006) for the finite element discretization and MOONoLith (Zulian 2016) for the intersection detection.

3. NUMERICAL VERIFICATION
We present three benchmarks to verify the TFSI approach presented in this contribution. The first benchmark is the Couette flow. For this experiment, we consider a one-way interaction where only the fluid is affected by the solid. The second benchmark consists of a fully coupled problem with two overlapping square domains, one representing the solid and the other the fluid. The third benchmark is used to analyze the behavior of heat transfer under different fluid flow regimes.

3.1 Couette flow
We consider a solid plate that moves horizontally with a constant speed of 0.05 m/s on top of the channel. The temperature of the solid is kept fixed ($T_s=283$ K). We prescribe no-slip boundary conditions and impose a temperature equal to 273 K at the bottom of the fluid channel. Details on the simulation set-up and the material properties are given in Figure 4 and Table 1, respectively. The penalty parameter is set equal to $10^{-6}$.

For visualization purposes, we analyze the two physics separately in Figure 5. In particular, Figure 5(a) shows the velocity fields of the fluid (on the right) and the solid (on the left) when steady-state conditions are achieved, whereas the temperature distribution is depicted in Figure 5(b). One may observe that both the velocity and the temperature constraints are properly enforced in the overlapping region.

**Table 1: Fluid and solid material properties**

<table>
<thead>
<tr>
<th>property</th>
<th>solid</th>
<th>fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>density ($\rho$) ($kg/m^3$)</td>
<td>2710</td>
<td>1000</td>
</tr>
<tr>
<td>viscosity ($\mu$) ($Pa \cdot s$)</td>
<td>-</td>
<td>1</td>
</tr>
<tr>
<td>heat capacity ($c$) ($J/(kg \cdot K)$)</td>
<td>921</td>
<td>4200</td>
</tr>
<tr>
<td>heat conductivity ($k$) ($W/(m \cdot K)$)</td>
<td>237</td>
<td>0.1</td>
</tr>
<tr>
<td>Young’s modulus ($E$) (GPa)</td>
<td>69</td>
<td>-</td>
</tr>
<tr>
<td>Poisson ratio ($\nu$)</td>
<td>0.3</td>
<td>-</td>
</tr>
</tbody>
</table>

**Figure 4: Set-up for the Couette flow benchmark.**

**Figure 5: Velocity (a) and temperature (b) of the Couette flow benchmark for the solid domain (left) and the fluid domain (right).**
Figure 6: Temperature profile across the flow channel. The x axis represents the distance from the bottom of the channel (meters).

Figure 6 presents a comparison between the numerical velocity profile analyzed in the middle of the fluid channel and the analytical solution. The good agreement between the two profiles shows the capability of our TFSI approach to enforce the equality of the discrete fields over the overlapping region in an accurate way without the need of any re-meshing tool.

3.2 Two-square blocks
A fully coupled strategy is verified through a benchmark (Pironkov et al. 2010) consisting of two square domains (Figure 7). The right square is a solid block, and the left square is occupied by the fluid. The left and right Dirichlet boundaries have constant temperature values, equal to $300\, K$ and $310\, K$, respectively. The fluid is modeled as stationary (we impose no-slip conditions on the entire boundary) and the solid is modeled as a rigid structure. The simulation setup and material properties are given in Figure 7 and Table 2, respectively. Due to the temperature gradient, heat is conducted along the axial direction until the thermodynamic equilibrium is reached. The analytical solutions for the steady-state temperature of the solid and fluid, respectively, are: $T_{\text{solid}} = 1.32x + 309.736, T_{\text{fluid}} = 98.68x + 300$, where $x$ is the axial coordinate starting from the left side of the fluid domain. Figure 8 depicts the steady-state temperature profiles in the axial direction at the mid-height of the two squares. The error computed by using the $L$-infinity norm is about $\sim 10^{-5}$. The good agreement between the numerical and the analytical solutions shows the capability of the overall TFSI framework to model complex scenarios where a fully coupled interaction between the solid and fluid phases is considered.

<table>
<thead>
<tr>
<th>property</th>
<th>solid</th>
<th>fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>density ($\rho$)$,(kg/m^3)$</td>
<td>7800</td>
<td>1.255</td>
</tr>
<tr>
<td>viscosity ($\mu$)$,(Pa\cdot s)$</td>
<td>- 1.8 $\times 10^{-5}$</td>
<td></td>
</tr>
<tr>
<td>heat capacity ($c$)$,(J/(kg\cdot k))$</td>
<td>450</td>
<td>1003</td>
</tr>
<tr>
<td>heat conductivity ($k$)$,(W/(m\cdot k))$</td>
<td>45</td>
<td>2.5 $\times 10^{-2}$</td>
</tr>
<tr>
<td>Young’s modulus ($E$)$,(GPa)$</td>
<td>2.1 $\times 10^{11}$</td>
<td>-</td>
</tr>
<tr>
<td>Poisson ratio ($\nu$)$,(-)$</td>
<td>0.3</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2: Fluid and solid material properties.

Figure 7: Set-up for the two-square blocks benchmark.
3.3 Flow regime study

We consider two obstacles embedded in a two-dimensional fluid channel where the fluid has an initial temperature equal to 550 $K$. We impose zero displacement at the bases of the two obstacles and a constant temperature equal to 300 $K$. The simulation setup and material properties for the solid and the fluid domains are given in Figure 9 and Table 3. The fluid has parabolic velocity at inlet defined as: $u_f = v_m (0.1 - y) y$, where $u_f$ is the fluid velocity, $y$ is the distance from the bottom of the channel.

![Figure 9: Set-up for the simulation of two obstacles in a flow-channel.](image)

We analyze three different flow regimes, slow, moderate, and fast, obtained by assigning to the parameter $v_m$ values equal to 5, 250, and 500 cm/s, respectively. Figure 10 shows the fluid velocity streamlines for all three scenarios at time $t = 0.20$ s. The fast fluid regime (Figure 10 bottom) reaches the maximum velocity of about 3.0 cm/s, at the center of the fracture aperture. Then, as the fluid reaches the wider section, the velocity decreases and finally stays constant close to the outlet. The same trend can be observed for both the slow and the moderate fluid regimes (Figure 10 top and Figure 10 middle, respectively), where a maximum velocity of about 1.6 cm/s and 0.03 cm/s is reached, respectively. Figure 11 shows the velocity profile at the inlet of the fluid channel for all the flow regimes. It can be observed that a parabolic profile is obtained in agreement with the Poiseuille flow model.

![Figure 8: Temperature profile along the axial direction evaluated at the mid-height of the two squares.](image)

\[ T_{\text{fluid}} = 98.68x + 300, \]
\[ T_{\text{solid}} = 1.32x + 309.736. \]

<table>
<thead>
<tr>
<th>Property</th>
<th>Solid</th>
<th>Fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density ($\rho$) (kg/m$^3$)</td>
<td>1200</td>
<td>1000</td>
</tr>
<tr>
<td>Viscosity ($\mu$) (Pa·s)</td>
<td>-</td>
<td>0.001</td>
</tr>
<tr>
<td>Heat capacity ($c$) (J/(kg·K))</td>
<td>790</td>
<td>4200</td>
</tr>
<tr>
<td>Heat conductivity ($k$) (W/(m·K))</td>
<td>3.2</td>
<td>0.6</td>
</tr>
<tr>
<td>Young’s modulus ($E$) (GPa)</td>
<td>$10^5$</td>
<td>-</td>
</tr>
<tr>
<td>Poisson ratio ($\nu$) (-)</td>
<td>0.4</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 3: Fluid and solid material properties.
Figure 10: Velocity streamlines (cm/s) at \( t = 0.20 \) s

Figure 11: Velocity profile at the cross section of the tunnel at \( x = 0.01 \) cm and \( t = 0.20 \) s.

Figure 12: Temperature profile at different time steps (\( t = 0.20 \) and \( 0.80 \) s) for (a) slow, (b) moderate, and (c) fast flow fluid.
In Figure 12 we plot snapshots of the spatial distribution of the fluid temperature at different time steps to study the effect of the fluid velocity on the heat transfer. Here, the left side depicts the temperature profiles at \( t = 0.20 \) s, whereas the right side shows the temperature profiled at \( t = 0.80 \) s. It can be seen that the conduction process is dominant at low fluid velocity values (Figures 12 top), whereas for high fluid velocities, heat transfer is driven by the advection phenomena (Figures 12 middle and Figure 12 bottom, respectively).

4. TFSI SIMULATION OF IDEALIZED ROCKS IMMERSED IN A FLOW FLOW-CHANNEL

In this section we show a proof of concept towards the application of our TFSI method for simulating Enhanced Geothermal Systems (EGS). The set-up consists of two-dimensional idealized fracture asperities, which are immersed in a channel-flow. Although this is a very simplified representation of the physical processes of EGS, it enables to demonstrate the modeling capabilities of the proposed TFSI framework relevant for geophysics applications. In particular, the circulation of cold fluid, which is then heated by the hot rock mass.

We consider a more complex geometry consisting of four idealized fracture asperities which are immersed in a channel-flow. A fluid is pumped with velocity \( v_m = 5 \text{ cm/s} \) and initial temperature equal to 323 \( K \). The rocks are clamped at their bases (i.e. zero-displacements are prescribed), and have an initial temperature equal to 373 \( K \). A constant temperature of 473 \( K \) is imposed on the top and bottom boundaries of the computational domain as depicted in Figure 13. The simulation set-up and material properties for the solid and the fluid domains are given in Figure 13 and Table 3. The fluid has a parabolic velocity at inlet defined as: \( u_f = v_m (1 - \gamma) y \), where \( u_f \) is the fluid velocity, \( y \) is the distance from the bottom of the channel, and \( v_m = 0.3 \text{ m/s} \).

![Figure 13: Set-up for the TFSI simulation of an idealized fracture geometry in a flow-channel.](image)

In Figure 15, we analyze the temperature profile at \( x = 2.0 \) m, i.e. on the right of the first fracture aperture, at different time instants (\( t = 1.0, 2.0, 3.0, 4.0 \) s). Due to the concurrent heating processes, i.e. conduction and advection, the temperature of the fluid progressively increases until it achieves a value equal to 420 \( K \). Such a trend can be also observed in Figure 16 where we show some snapshots of both the temperature and the fluid velocity in the flow channel. Here, we can see that the fluid velocity field increases in the narrow parts of the fracture. As a consequence, despite the low velocity profile prescribed at the inlet (Figure 14), the heat transfer is driven from both the diffusive and the advection phenomena, which makes this scenario more similar to the one analyzed in Figure 12 (middle row). On the contrary, close to the entrance the observed behavior is more similar to the one analyzed in Figure 12 (top row). Indeed, due to the small values of the fluid velocity field, the heat transfer is mostly driven by conduction and leads the fluid temperature to progressively increases before moving through the fracture asperities.

![Figure 14: Velocity profile at the cross section of the tunnel at \( x = 0.01 \) cm and \( t = 0.20 \) s.](image)
5. CONCLUSION

In this paper, we presented a framework for the numerical simulation of fully coupled thermo-hydro-mechanical interaction based on the fictitious domain method. This approach simplifies the setup of TFSI simulations and allows us to solve coupled problems without an explicit representation of the complex boundary between two domains. First, we define the governing equations of linear thermo-elasticity to describe the heat transfer in a hot rock mass. Then, Navier-Stokes and energy equations are adopted to model the thermal phenomena in an incompressible Newtonian fluid. Finally, the two problems are coupled in a staggered manner by means of a penalty method based on the variational transfer. We verified the presented formulations with two-dimensional benchmarks on simple geometries. Then, we demonstrated that our methodology is capable of simulating the behavior of fluid flow and heat transfer in idealized two-dimensional fracture geometries. In summary, this article illustrates a proof of concept for the simulation of thermal-fluid-structure interaction in geothermal applications. Further investigations and numerical experiments will involve more complex three-dimensional geometries also including rough fracture surfaces.
Figure 16: Snapshots of the simulation at different time instants of the temperature (left) and the velocity field (right).

REFERENCES


Hassaniankhoshkroud et al.


