Non-linearities and Upscaling in Porous Media

Fluid-Porous-Media Interaction Modeling with Staggered Solution Schemes

Asgharzadeh, P.

Supervisors:
S. Zinatbakhsh, M.Sc.
Dipl.-Ing. D. Koch

Preprint 2015/10
# Contents

1 Introduction .......................... 1  
   1.1 Motivation ............................ 1  
   1.2 Fluid-porous-media interaction ........ 3  
   1.3 Structure of the thesis ................. 4  

2 Theoretical Basics .................... 5  
   2.1 The Theory of Porous Media ............. 5  
   2.2 Kinematics of multi-phase continua ....... 6  
      2.2.1 Basic definitions ................. 7  
   2.3 Balance relations ..................... 8  
      2.3.1 Balance of mass ................... 9  
      2.3.2 Balance of momentum .............. 9  
   2.4 Constitutive equations ............... 10  
   2.5 Fluid representation ................. 12  
      2.5.1 Modified Eulerian description of fluid behavior ........ 12  
      2.5.2 Bulk fluid momentum balance ........ 14  
      2.5.3 Constitutive equations for bulk fluid representation . . 15  

3 Implementation of the Biphasic Porous Material and the Bulk Fluid Solvers in PANDAS 17  
   3.1 IBVP of the dynamic representation of porous media ........... 17  
      3.1.1 Governing weak formulations of biphasic porous medium .... 18  
      3.1.2 Time discretization of the coupled problem .............. 19  
   3.2 IBVP of fluid dynamics ................ 20  
      3.2.1 Governing weak formulations of the bulk fluid .......... 22  
      3.2.2 Numerical treatment of the Navier-Stokes equation ........ 22  
   3.3 Spatial discretization ................ 23  

4 Fluid-Porous-Media Coupling ............ 25  
   4.1 Partitioned solution procedure ........... 26  
   4.2 The conventional serial staggered algorithm ............... 26  
   4.3 Coupling two PANDAS solvers ............ 28
4.3.1 Implementation of CSS for two PANDAS solvers .......................... 29
4.3.2 Communication between the solvers ........................................ 30
4.4 Detailed algorithm ......................................................................... 31
4.5 Translation of the output to the boundary conditions ....................... 32

5 Numerical Examples ................................................................. 35
5.1 Dynamic representation of biphasic porous material .......................... 35
5.2 Dynamic fluid flow ...................................................................... 37
5.2.1 Dynamic flow of viscous incompressible fluid ............................... 38
5.2.2 Inviscid fluid flow model with moving boundaries ......................... 41
5.3 Numerical example of the FPMI problem ....................................... 44
5.3.1 Conditional stability of CSS ..................................................... 50

6 Outlook ...................................................................................... 55

Conclusion and Outlook .................................................................. 55

Bibliography .................................................................................. 59
# List of Figures

1.1 Schematic view of the blood-vein interaction ........................................ 3
1.2 Fracking process as an example of fluid-porous-media interaction ............ 4
1.3 Water-soil interaction in a geothermal power plant ............................. 4

2.1 Kinematics of a biphasic mixture ......................................................... 7
2.2 *Lagrangian, Eulerian* and ALE representation of fluid behavior ............ 13

4.1 CSS : the conventional serial staggered procedure .............................. 27
4.2 CSS : the conventional serial staggered procedure with fluid sub-cycling ... 28
4.3 Coupling process flowchart ................................................................. 30
4.4 Updating boundary conditions and geometry ....................................... 31

5.1 Geometry and spatial discretization for the biphasic example .................. 35
5.2 Boundary conditions and loading path of dynamic compression of the biphasic saturated column ............................................................. 36
5.3 Solid skeleton displacement of top part of column through time .............. 37
5.4 Seepage velocity on top of the column through time ............................ 37
5.5 Geometry and spatial discretization of the viscous fluid flow model ......... 38
5.6 Boundary conditions and loading path of dynamic movement of viscous fluid flow inside a pipe ................................................................. 39
5.7 Fluid velocity profile through $x$ direction at $t = 2S$ ............................ 40
5.8 Fluid velocity profile through $x$ direction at $t = 5S$ ............................ 40
5.9 Fluid velocity profile through the whole column at $t = 2S$. The color range in the image starts from red= 0 and gets to blue$> 0$. .............. 41
5.10 Boundary conditions and moving wall .................................................. 42
5.11 Fluid velocity in the $y$-direction when the wall is moving toward left at $t = 1S$. The color range presented in the image starts from blue$< 0$ to red$> 0$. ................................................................. 42
5.12 Fluid velocity in the $y$-direction when the wall is moving toward right at $t = 1.2xS$. The color range presented in the image starts from blue$< 0$ to red$> 0$. ................................................................. 43
5.13 Geometry and boundary conditions of the numerical FPMI example ......... 44
5.14 Snapshots of the behavior of each of the subsystems in selected times as $t = 0.01S$, $t = 0.1S$, and $t = 1.0S$. The blue vectors show the bulk and pore fluid velocities. Pressure is represented by the color range in the background ........................................... 47
5.15 The velocity of bulk fluid in \( y \)-direction at the interface at points \( A \) and \( B \) ................................. 48
5.16 Pressure vs. time at the interface at points \( A \) and \( B \). .................. 49
5.17 Comparing the results of the CSS method with the results of the LLM method. ......................................................... 50
5.18 Magnified deformation of the mesh in the bulk fluid and porous material 51
5.19 Results of the model with \( \Delta t = 10^{-2}S, 10^{-3}, \) and \( 10^{-4}S \) .................. 52
5.20 Comparison of the results for \( \Delta t = 10^{-3} S \) and \( \Delta t = 10^{-4} S \). .......... 53
Chapter 1: Introduction

1.1 Motivation

In this work, a partitioned scheme for numerical simulation of the surface-coupled problem of fluid-porous-media interaction (FPMI) has been proposed. To this end, the conventional serial staggered (CSS) procedure of Farhat & Lesoinne [10] is adopted in a way that facilitates an automatic spatial partitioning of the problem, and a serial treatment of the interacting components. Moreover, proceeding from the interaction between an incompressible bulk fluid with a saturated biphasic porous medium with intrinsically incompressible and inert constituents, the characteristics of the governing equations are scrutinized and the various constraints within the subsystems are identified.

Under the expression fluid-porous-media interaction, we define a surface interaction among several non-overlapping subsystems composed of either a bulk fluid or a porous medium. This could be interaction of blood with a blood vessel, a body of water with an earth dam structure, water flow through cracks during fracking process, or acoustic waves with acoustic panels used in soundproofing, etc. The mathematical model of such a phenomenon comprises a coupled differential algebraic equation (DAE) system in space and time, including the governing equations of the subsystems subjected to specific consistency constraints at the interface. The solution of these equations reveals the behavior of the system in different circumstances specified by the initial and boundary conditions.

In order to solve this DAE system numerically, one can either follow a monolithic or a staggered approach. The conventional serial staggered algorithm (CSS) and the local Lagrange multipliers method (LLM) proposed by Park & Felippa [12] are two of the most popular examples of the partitioned solution schemes. Furthermore, a common problem in case of using numerical solutions for the representation of the incompressible fluid behavior, i.e., Navier-Stokes equation, is oscillations in the pressure field which happens due to the fact that the pressure term is not directly present in the incompressibility constraint. In this work, the mentioned issue has been prevented by the artificial compressibility method of Chorin [4]. Using the artificial compressibility formulation, one relaxes the problem by allowing for some violations of the constraints addressing the continuity in each subsystem. This procedure will also transform the set of differential equations (DE) to the hyperbolic form which makes the numerical treatment easier.

In conclusion, combining the artificial compressibility method formulation of the subsystems with CSS method for partitioning of the domain yields a promising can-
didate for decoupled solution of the FPMI problem. To implement this solution strategy for modeling the fluid-porous-media interaction is the main purpose of this contribution.

For implementation of this solution, Porous media Adaptive Nonlinear finite element solver based on Differential Algebraic System (PANDAS) will be used, which is especially designed for solution of strongly coupled multi-phasic-porous-media problems. In order to do so, two different material routines for the inviscid and incompressible fluid and for the biphasic saturated porous medium will be defined. In addition, another environment will be used to couple these two solvers together. The whole process will be managed by a main code, which will run two different PANDAS solvers and then the results will be sent back and forth between these two solvers at each time step and the result of one subsystem (e.g. velocity of the bulk fluid at the interface) will be used as the boundary conditions for the other subsystem. Then for the next time step, the main code will update the two problems based on the results gained from previous time step from each subsystem. This process will be experienced for each subsystem at each sub-domain through the required time.

On the bulk fluid subsystem, the balance equations will be solved for the fluid velocity with the help of modified Eulerian description (MED). The MED would make it possible to move the nodes on the boundary at the interface for fluid subsystem in each step. This is necessary in order to have continuity in the simulation (no gap between the bulk fluid and the porous medium). Utilization of the MED for representation of the fluid behavior is implemented as a material routine in PANDAS and the velocity of the fluid at the nodes at the interface will be determined. For solid skeleton in the saturated porous medium, which is governed by the Theory of Porous Media (TPM) [7, 8], the Lagrangean description will be used. Furthermore, for the pore fluid, the modified Eulerian description will be used via the seepage velocity vector denoting the fluid motion relative to the deforming solid skeleton. Utilizing finite-element method and also applying the proper boundary conditions, the response of the porous medium and pore fluid constituents will be determined and the boundary conditions at the interface will be updated at each time step for the solvers. After requested amount of steps, the model would represent a simulation of the fluid-porous-media interaction.
1.2 Fluid-porous-media interaction

Fluid-porous-media interaction is a phenomenon occurring in a vast variety of natural environments ranging from microscopic point of view in biological tissues to macroscopic level in water-soil interaction in nature. The ability to model this physical incident brings about more realistic simulations regarding the problems with the fluid and porous medium contributions.

The current knowledge of the behavior of porous media enables one to have a close to sufficient understanding of biological materials, which are one of the most important environments consisting of porous media. Having this in mind that all biological tissues are actively under change and their responses vary due to their interactions, understanding the circumstances produced by these interactions facilitates the completion of perception toward biological tissues behavior. One outstanding example is the blood-vein interactions, cf. Figure 1.1.

![Figure 1.1: Schematic view of the blood-vein interaction.](image)

Considering the vein as a porous medium holding the viscous fluid inside, with help of FPMI modeling physical interactions between these two media gets more clear. It can lead to further investigation regarding biological researches.

Another important occurrence of fluid-porous-media interaction is water-soil interaction. This phenomenon exists in different environments, e.g., underground water resorts, rivers and floods. In addition, in human-made structures such as soil dams, fracking facilities and geothermal power plants, the same physical contributions of FPMI happens, cf. Figure 1.2 and Figure 1.3.
1.3 Structure of the thesis

In Chapter 2 the fundamental concepts of continuum mechanics and the TPM was used to develop the governing equations for the porous medium and the bulk fluid. Chapter 3 is about the explanation of the process of implementation of biphasic material routine which used for the simulation of the saturated porous media. Furthermore, dynamic fluid model used for the representation of the bulk fluid behavior in PANDAS is discussed. In Chapter 4, the coupling procedure of two PANDAS solvers are explored, and in Chapter 5 four different numerical examples are presented. Finally, in Chapter 6, the conclusions about the overall results of this work are discussed and the outlines of this project are explored.
Chapter 2: Theoretical Basics

In this chapter the description of saturated porous media and bulk fluid in macroscopic scale is explored via the introduction of multiphasic continuum theories. To do so, one needs to start with understanding the fundamental concepts of the TPM, which will be accompanied by utilization of the kinematic relations of the multiphasic media and the balance relations. Furthermore, the concept of the MED for fluid representation is discussed. The mathematical modeling is completed in Chapter 3 by introducing thermodynamically consistent constitutive relations.

2.1 The Theory of Porous Media

In order to model the microscopic behavior of a biphasic porous medium consisting of a porous solid skeleton saturated by a single interstitial fluid, one can use the powerful framework of the TPM. In this regard, a representative elementary volume (REV) will replace the solid and the fluid aggregate with a random granular geometry in a state of ideal disarrangement, cf. Ehlers [7] and Bowen [2].

A smeared out continuum $\varphi$ with overlapped, interacting and statistically distributed solid and fluid aggregates $\varphi^\alpha (\alpha = S: \text{solid phase}; \alpha = F: \text{pore-fluid phase})$ is the result of a homogenization process performed on the REV. Thus, at any given macroscopic subspace, the following relation holds:

$$\varphi = \bigcup \varphi^\alpha = \varphi^S \cup \varphi^F.$$  \hspace{1cm} (2.1)

The homogenization process takes place by the integration of the microscopic information for each constituent. Furthermore, the concept of volume fractions, which is fundamental to the Theory of Porous Media, is introduced, as it is shown in equation 2.2:

$$n^\alpha = \frac{\text{d}v^\alpha}{\text{d}v}. \hspace{1cm} (2.2)$$

This is the result of the assumption of immiscible aggregates and extension of the Mixture Theory. By performing a volumetric averaging process of all interrelated constituents over the REV, one will proceed with the homogenization goal and the incorporated physical fields of the microstructure are represented by their volume proportions on the macroscopic level.

In order to have the overall volume $V$ of the homogenized, fluid-saturated porous body $\mathcal{B}$ within the TPM, the concept of volume fractions is used and the sum of the
Chapter 2: Theoretical Basics

partial volumes $V^\alpha$ of the constituents is calculated as

$$V = \int_B dv = \sum_\alpha V^\alpha \quad \text{with} \quad V^\alpha = \int_B dv^\alpha =: \int_B n^\alpha dv$$ (2.3)

With this in mind, the volume fraction $n^\alpha$ of a constituent $\varphi^\alpha$ is defined as the ratio of space occupied by the constituent $\varphi^\alpha$ of the whole REV with $dv^\alpha$ and $dv$ being the constituent and the bulk volume elements, respectively.

Furthermore, in case of having the saturation condition, it reads,

$$\sum_\alpha n^\alpha = n^S + n^F = 1.$$ (2.4)

In the current treatment of multiphasic materials, it is always assumed that $1 > n^\alpha > 0$. Therefore, the convergence of the solution to pure solid or pure fluid from the following sets of formulation is not the case of study here.

Distinct density functions for each constituent are other parameters that are necessary for the formulation of balance relations for the multiphasic porous media. These can be derived from the definition of volume fraction and underlying concept of density. Moreover, two different kinds of density functions for each constituent can be specified: These are the material (or effective) density function $\rho^{\alpha R}$ relating the local mass $dm^\alpha$ to the partial volume element $dv^\alpha$, and the partial density function $\rho^\alpha$ relating $dm^\alpha$ to the bulk volume element $dv$:

$$\rho^{\alpha R} := \frac{dm^\alpha}{dv^\alpha}, \quad \rho^\alpha := \frac{dm^\alpha}{dv}.$$ (2.5)

The density of the overall aggregate $\rho$ can be calculated by summing up the partial densities of the constituents and the partial and material densities are related via $n^\alpha$. Moreover, the material incompressibility of the constituents would result in $\rho^{\alpha R} = \text{const.}$ but this does not lead to incompressibility of the overall aggregate. $\rho^\alpha = n^\alpha \rho^{\alpha R}$ shows that by change of volume fractions of the constituents (e.g. drainage) the density of the overall aggregate can change.

2.2 Kinematics of multi-phase continua

Continuum mechanics of single-phase materials is the main inspiration of the kinematic formulations in theory of porous media. By performing required modifications, Ehlers [6], one will end up with the description of the motion of multiphasic continua. In this section, a brief overview of the kinematic relations of multiphasic continuum mechanics which are used for the treatment of the considered biphasic material in small deformation regime is presented.
2.2 Kinematics of multi-phase continua

2.2.1 Basic definitions

According to the concept of superimposed continua with internal interactions and individual states of motion, there is a unique motion function for each constituent $\varphi^\alpha$ starting from positions $X_\alpha$ in the reference configuration at the starting time $t_0$. Each spatial point of the body $B$ at the actual time $t$ is occupied by particles of all constituents $\varphi^\alpha$, cf. Figure 2.1. Accordingly the vector-valued field functions of motion, velocity and acceleration of each constituent are defined as follows:

\[
x = \chi_\alpha (X_\alpha, t), \quad \dot{x}_\alpha := \frac{\partial \chi_\alpha (X_\alpha, t)}{\partial t}, \quad \ddot{x}_\alpha := \frac{\partial^2 \chi_\alpha (X_\alpha, t)}{\partial t^2}.
\]

(2.6)

In this context, unique motion (mapping) functions $\chi_\alpha$ and the material description of the current position of each constituent is used.

Next, the material time derivatives of the motion function are defined in equation (2.6). Wherein, $(\cdot)'_\alpha$ and $(\cdot)''_\alpha$ denote the first and second material time derivatives of the motion function $\chi_\alpha$. Furthermore, $(\cdot)'_\alpha$ shows the material time derivative with respect to the motion of the constituent $\varphi^\alpha$, such that

\[
(\cdot)'_\alpha := \frac{d_\alpha (\cdot)}{dt} = \frac{\partial (\cdot)}{\partial t} + \text{grad}(\cdot) v_\alpha.
\]

(2.7)

Furthermore, the requirement for a unique and uniquely invertible motion function
Chapter 2: Theoretical Basics

is

\[ X_\alpha = \chi^{-1}_\alpha (x, t) \quad \text{if} \quad J_\alpha := \text{det} \left( \frac{\partial \chi_\alpha}{\partial X_\alpha} \right) \neq 0. \quad (2.8) \]

In this regard, the material deformation gradient \( F_\alpha \) and its inverse \( F^{-1}_\alpha \) are given by

\[ F_\alpha = \frac{\partial \chi_\alpha}{\partial X_\alpha} = \text{Grad}_\alpha x \quad \text{and} \quad F^{-1}_\alpha = \frac{\partial \chi^{-1}_\alpha}{\partial x} = \text{grad} X_\alpha \quad (2.9) \]

with the gradient operator \( \text{Grad}_\alpha (\cdot) \) as a representation for the partial derivative with respect to the reference position vector \( X_\alpha \), and \( \text{grad} (\cdot) \) as the partial derivative with respect to the actual position vector \( x \).

In the framework of solid-fluid coupled problems, the solid motion is usually expressed in a Lagrangean description via the displacement vector \( u_S \), and the fluid is considered using a modified Eulerian description via the seepage velocity \( w_F \) as

\[ w_F := v_F - v_S, \quad (2.10) \]

where, \( v_F \) and \( v_S \) are the velocities of the fluid and the solid phases, respectively.

2.3 Balance relations

Balance equations including the balance of mass, linear momentum, momentum of momentum and energy are the expressions that continuum mechanics builds its foundation upon them. Independent of material constitution, these relations are valid for all continuum bodies. The notions of mass, energy and force are the equivalence of primary concepts of time, position and velocity in kinematics in the framework of the classic mechanics of continua. These entities can be axiomatically explored via master balances, which give a frame for all balances. To do so, Truesdell’s metaphysical principles [16], which extend the classical balance laws, are used. These principles are valid for the multiphasic materials, in which the interactions between the constituents are taken into account, and they read as follows:

1. All properties of the mixture must be mathematical consequences of properties of the constituents.

2. So as to describe the motion of a constituent, we may in imagination isolate it from the rest of the mixture, provided we allow properly for the actions of the other constituents upon it.

3. The motion of a mixture is governed by the same equations as is a single body.
Additionally, in this work, following assumptions are used to simplify the formulation of the governing equations in the global form:

- Mass exchange between the constituents is excluded and mass production terms $\dot{\rho}^\alpha$ are neglected.
- Material incompressibility of the constituents is assumed, i.e., $\rho^{\alpha R} = \text{const.}$
- Constant temperature for all constituents is considered, i.e., $\Theta^\alpha = \Theta = \text{const.}$
- Gravitation forces are neglected, i.e., $g \approx 0$. 

### 2.3.1 Balance of mass

In case of a single body $B$ with no mass transfer through its surface $S$, the mass of the body remains conserved through the deformation process. The same concept is valid in case of a multiphasic body and it is applied to the overall aggregate and to the constituents. Here, incorporating the specific balance terms into the local master balances yields the local mass balances (continuity equations) of the overall body

$$\dot{\rho} + \rho \text{div} \dot{x} = 0, \quad (2.11)$$

and the constituents

$$(\rho^\alpha)' + \rho^\alpha \text{div} v^\alpha = \dot{\rho}^\alpha. \quad (2.12)$$

Under the first two assumptions, the local mass balances of the solid and fluid constituents transform into volume balances. The volume balances of the overall aggregate is obtained after summing up the local volume balances of its constituents yielding:

$$\begin{cases}
(n^S)'_S + n^S \text{div} \dot{x}_S = 0 \\
(n^F)'_F + n^F \text{div} \dot{x}_F = 0
\end{cases} \quad \rightarrow \quad \text{div}(v_S + n^F w_F) = 0. \quad (2.13)$$

### 2.3.2 Balance of momentum

According to classical Newtonian mechanics laws, the sum of the forces acting on a body $B$ is equal to changes of the linear momentum of the body over the time. In multiphasic continua, as a result of the homogenization, the multiphasic body is
treated as singlephasic material. The forces affecting the multiphasic body can be divided into two groups based on the way they interact with the body. Following this categorization, the two different types of acting forces are as follows: the mass-specific body force $b$ as a distant effect of the surrounding bodies, and the surface traction at the vicinity represented by the Caushy stress tensor $T$. The overall momentum balance reads

$$\rho \ddot{x} = \text{div } T + \rho \ b.$$  \hfill (2.14)

Moreover, the constituent momentum balance reads

$$\rho^\alpha (v_\alpha)' = \text{div } T^\alpha + \rho^\alpha b^\alpha + \hat{\rho}^\alpha.$$  \hfill (2.15)

Following the first and the two last assumptions expressed in section 2.3 and also assuming that the fluid acceleration is almost identical to the solid acceleration (for low and moderate frequencies, say below 30Hz), the momentum balance of the overall aggregate is obtained from the sum of the local momentum balances of the solid and fluid constituents as

$$\text{div } T^S + \hat{\rho}^S - \rho^S \dot{v}_S = 0 \Bigg\} \rightarrow \text{div}(T^S + T^F) = 0.$$  \hfill (2.16)

Therefore, the overall Caushy stress tensor is the sum of the partial Caushy stresses of the solid and the pore fluid, and $\hat{\rho}^S = -\hat{\rho}^F$ is the momentum production term.

### 2.4 Constitutive equations

Next to the local balance equations, the physical response of the biphasic continua is described by means of constitutive equations. In this context, the presented formulations are related to the partial Caushy stress tensors of the solid and pore fluid, and the momentum production term. The constitutive relations which has been used in this work to solve the state equations of a biphasic porous medium are as follows:

$$T^S_E = n^S p^F I + T^S,$$

$$0 = n^F p^F I + T^F,$$  \hfill (2.17)

$$\hat{\rho}^F = -\frac{(n^F)^2 \gamma F R}{k^F} w_F + p^F \text{ grad } n^F.$$
In the previous equation, it is assumed that extra stress $T^F_E$ value for pore bulk fluid is zero due to the fact that it is much smaller than the momentum production term in the momentum balance equation. Furthermore, $p^F$ is the pore pressure, $\gamma^{FR}$ is the effective weight of the pore fluid and $k^F$ represents the Darcy permeability (hydraulic conductivity).

Furthermore, $T^S_E$ in the geometrically linear regime is determined by the Hookean elasticity law

$$T^S_E = 2\mu^S\varepsilon + \lambda^S(\varepsilon_S \cdot I)I,$$  \hspace{1cm} (2.18)

where $\varepsilon_S := \text{sym} (\text{grad} u_S)$ is the linear solid strain tensor, and $\mu^S$ and $\lambda^S$ are the Lamé constants of the porous solid matrix.

Based on the TPM, the pore fluid motion is commonly described in the modified Eulerian framework. Thus, the material time derivative with respect to the fluid motion in momentum balance relation can be replaced via the following relation derived from equations (2.7) and (2.10):

$$(\cdot)'_F = (\cdot)'_S + \text{grad}(\cdot)w_F. \hspace{1cm} (2.19)$$

Thus after inserting (2.17) and by including (2.19), the complete set of governing equations for the porous medium subsystem is as follow

$$\begin{align*}
(u_S)'_S &= v_S, \hspace{1cm} (2.20) \\
\rho^S (v_S)'_S &= \text{div} T^S_E - n^S \text{grad} p^F + \frac{(n^F)^2 \gamma^{FR}}{k^F} w_F, \hspace{1cm} (2.21) \\
p^F [ (v_F)'_S + (\text{grad} v_F) w_F ] &= -n^F \text{grad} p^F - \frac{(n^F)^2 \gamma^{FR}}{k^F} w_F, \hspace{1cm} (2.22) \\
0 &= \text{div} (v_S + n^F w_F). \hspace{1cm} (2.23)
\end{align*}$$

In this context, it is worth mentioning that the general three-field description in some cases may be boiled down to a special bivariate formulation. A simpler consolidation model based on the primary unknowns $u_S$ and $p^F$, thereby accepting some additional error. In this case, the fluid momentum balance (2.22) just serves as an explicit equation for the filter velocity $n^F w_F$, which is then substituted into the other balances. In doing so, (2.21) and (2.23) are replaced by:

- Modified solid (or mixture) momentum balance::

$$\begin{align*}
(\rho^S + \rho^F)(v_S)'_S &= \text{div} (T^S_E - p^F I). \hspace{1cm} (2.24)
\end{align*}$$
• Modified mixture volume balance:

\[
\text{div}\left( \mathbf{v}_S - \frac{k^F}{g} [(\mathbf{v}_S)_S'] \right) - \frac{k^F}{\gamma^F_R} \Delta p^F = 0.
\] (2.25)

Therein, \( \Delta(\cdot) = \text{div} \text{grad}(\cdot) \) denotes the Laplace operator and the convective term and the porosity gradient were neglected because of our linear analysis.

### 2.5 Fluid representation

The set of differential equations consisting of the fluid linear-momentum balance and continuity condition describe the behavior of an incompressible and inviscid fluid. The solution to this set of equations gives the information of the movement of the fluid (fluid velocity profile) through the time. Therefore, one of traditional computational fluid dynamic (CFD) problems is to handle this set of equations numerically. With these two equations at hand, one can describe the behavior of fluid part in the fluid-porous-media interaction problems

\[
\chi^L : \begin{cases} 
\rho^L (\mathbf{v}_L)_L' = -\text{grad} p^L, \\
0 = \text{div } \mathbf{v}_L.
\end{cases}
\] (2.26)

Therein, \( \rho^L \) is the density, \( \mathbf{v}_L \) represents the velocity, and \( p^L \) is the hydraulic pressure of the bulk fluid.

#### 2.5.1 Modified Eulerian description of fluid behavior

One major concern with the solution of the equation (2.26) is that it represents the behavior of the fluid in Lagrangean frame as if the particles of the fluid are attached to the mesh (or vise versa) which means the observer is moving with the mesh. The other option is to use the Eulerian representation, which would result in an stationary observer. In order to transform equation (2.26) to the Eulerian version, one must use (2.19) instead of the material time derivative which yields

\[
\rho^L \left[ \frac{\partial \mathbf{v}_L}{\partial t} + (\text{grad} \mathbf{v}_L)\mathbf{v}_L \right] = -\text{grad } p^L.
\] (2.27)

As it is obvious in equation (2.27), this version of representation, though it is common in commercial CFD softwares, is not suitable to solve problems with moving boundaries, which is the case of FPMI. An alternative representation to serve the purpose of fluid dynamic representation with moving boundaries is the so-called ar-
2.5 Fluid representation

Arbitrary Lagrange-Eulerean (ALE) methods, Donea et al. [5], where the motion of the fluid is described by an observer (a mesh) with arbitrary motion. Alternatively, using the concept of the modified Eulerean description (MED) yields a similar, yet not identical, formulation to the ALE method, cf. Figure 2.2.

![Diagram showing Lagrangean, Eulerean, and ALE representation of fluid behavior](image)

**Figure 2.2:** Lagrangean, Eulerean and ALE representation of fluid behavior

The ideal description of the bulk fluid in case of the FPMI problem would be representing the motion of the fluid with respect to the motion of a mesh (from the viewpoint of the observer sitting on the mesh) which is attached to the moving boundaries. This approach toward the representation of the bulk fluid motion completely matches the concept of the MED, which is also used in the framework of TPM. In case of TPM, through the MED one can describe the motion of the pore fluid with respect to the motion of the solid skeleton. In this regard, considering an exemplary constituent \( \varphi^\beta \) as the material whose motion should be described, we firstly introduce the mesh as an additional constituent-like \( \varphi^M \) to the domain. Thus, assuming \( \varphi^\beta \) to be a continuum occupying the whole domain, one can assign a material particle \( \mathcal{P}^\beta \) with the actual position vector \( \mathbf{x}_\beta \) to each mesh-point \( \mathcal{P}^M \) with the actual position vector \( \mathbf{x}_M \), such that

\[
\mathbf{x}_\beta = \mathbf{x}_M =: \mathbf{x}.
\]  

(2.28)

Furthermore, proceeding from the individual unique and uniquely invertible motion
functions $\chi_{\beta}$ and $\chi_{M}$ yields

$$\begin{align*}
x &= \chi_{\beta}(X_{\beta}, t) & \leftrightarrow & & X_{\beta} &= \chi_{\beta}^{-1}(x, t), \\
x &= \chi_{M}(X_{M}, t) & \leftrightarrow & & X_{M} &= \chi_{M}^{-1}(x, t).
\end{align*}$$

(2.29)

Analogously, as it was done in biphasic medium representation via the TPM, the material derivate of the mechanical quantities in the framework of the MED is given by

$$\begin{align*}
(\cdot)'_{\beta} &= (\cdot)'_{M} + \text{grad}(\cdot)(v_{\beta} - v_{M}).
\end{align*}$$

(2.30)

### 2.5.2 Bulk fluid momentum balance

Employing the results from previous chapter allows for a general representation of the thermo-mechanical balance relations. The generality is such that the Lagrangean and the Eulerian representations of the relations become special cases, which could automatically be achieved by an appropriate choice of the mesh velocity. To demonstrate this, consider the linear momentum balance of the bulk fluid $\varphi_{L}$ as given by (2.26). Replacing $(v_{L})'_{L}$ by means of (2.30) yields

$$\rho_{L}[(v_{L})'_{M} + (\text{grad } v_{L})w_{L}] = -\text{grad } p_{L}.$$  

(2.31)

Wherein, $w_{L} := v_{L} - v_{M}$.

$$\begin{align*}
\begin{cases}
v_{M} \equiv v_{L} & \rightarrow & \rho_{L}(v_{L})'_{M} \equiv \rho_{L}(v_{L})'_{L} = -\text{grad } p_{L} & : \text{Lagrangian rep.} \\
v_{M} \equiv 0 & \rightarrow & \rho_{L}\left[\frac{\partial v_{L}}{\partial t} + (\text{grad } v_{L})v_{L}\right] = -\text{grad } p_{L} & : \text{Eulerean rep.}
\end{cases}
\end{align*}$$

(2.32)

Based on the arbitrariness of the motion of the mesh and the fact that some parts of the mesh are attached to the moving boundaries one need to solve the motion equations of the mesh with the prescribed boundary conditions to determine the motion of the mesh inside the domain. It can be done by considering $\varphi_{M}$ as a pseudo-material with unit density, the motion of which is governed by

$$\begin{align*}
\begin{cases}
(u_{M})'_{M} = v_{M}, \\
(v_{M})'_{M} = \text{div } T_{M}.
\end{cases}
\end{align*}$$

(2.33)
2.5 Fluid representation

Herein, although the mesh is considered as a pseudo material, momentum balance (with required modifications such as removing the external forces) holds for the motion of the mesh. Therefore, by imagining a type of behavior for the mesh, such as linear behavior, one would subscribe to the specified behavior of the mesh and, therefore, the stress tensor $T^M$ can be determined

$$T^M = 2\mu^M \varepsilon_M + \lambda^M (\varepsilon_M \cdot I) I.$$  \hfill (2.34)

Note that $\varphi^M$ plays the role of a fictitious material, which neither interacts with the bulk fluid nor can carry any applied boundary load. Hence, (2.33) can be solved separately from and prior to (2.31) and only in conjunction with a Dirichlet boundary condition. See Chapter 3.

2.5.3 Constitutive equations for bulk fluid representation

Constitutive relations for stress in the bulk fluid depend on what type of fluid behavior is being represented in the model. In this work, two types of fluid behavior was considered, which are viscous fluid and inviscid fluid. In the case of inviscid fluid, due to the fact that the friction in fluid particles is very small the extra stress term $T^E$ can be neglected. Therefore, for the total stress, one will have the following relation

$$T^L = -p^L I.$$ \hfill (2.35)

In the case of a viscous fluid, the extra stress term $T^E$ is not negligible anymore and a form of the constitutive equation for it would be needed as well

$$T^E = 2\mu^L D_L,$$

$$D^L = \text{sym} (\text{grad} L_L),$$

$$L^L := \text{grad} v_L.$$ \hfill (2.36)

In this equation, $D_L$ is the symmetric part of the fluid spatial velocity gradient and $\mu^L$ is the fluid dynamic viscosity. If that is the case, the momentum balance of the bulk fluid would have the following form:

$$\rho^L \left[ (v_L)' + (\text{grad} v_L) w_L \right] = \text{div} T^E - \text{grad} p^L.$$ \hfill (2.37)
Chapter 3: Implementation of the Biphasic Porous Material and the Bulk Fluid Solvers in PANDAS

Proceeding from the specified balance relations from Chapter 2, in this section the different methods that were used for the implementation of the governing sets of coupled balance relations for the biphasic porous media and the balance equations for the bulk fluid will be discussed.

In order to solve the initial-boundary-value problems (IBVP) of fluid-porous-media interaction, Finite-Element Method (FEM) was used for the spatial discretization, and based on the chosen primary values, required weak forms of the balance equations were implemented in two different sets of PANDAS material routines.

3.1 IBVP of the dynamic representation of porous media

The differential equations which represent the strong or continuous formulations with additional restraints in time and space, have the following elements:

- **Primary variables** $u(x, t)$: In both problems of biphasic porous media and bulk fluid, a set of major unknowns of the system of equations under investigations were chosen as
  \[ u^P(x, t) = [u_S, v_F, p^F], \]  
  which represent the displacement of the solid skeleton, pore fluid velocity and pore fluid pressure for the porous medium respectively.

- **Secondary variables**: These are functions of the primary variables that in our case consist of $v_S, w_F, \varepsilon_S$, and $n^S$ which are solid skeleton velocity (mesh velocity in case of bulk fluid), seepage velocity and deformation dependent solid volume fraction.

- **Initial conditions** ($t = t_0, \forall x \in \Omega$)

- **Boundary conditions** ($t \in [0, T], x \in \Gamma$): The surface $\Gamma = \partial \Omega$ is split into Dirichlet (essential) and Neumann (natural) boundaries allowing for application of the different external boundary conditions based on the work of Markert.
In particular, these boundaries for the given set of equations can be defined in general form as follows:

\[
\text{\textit{Dirichlet}} \quad u_S(x, t) = \bar{u}_S(x, t) \quad x \in \Gamma_{us} \\
\text{\textit{Neumann}} \quad t^S(x, t) = \bar{t}^S(x, t) = T^S n \quad x \in \Gamma_{ts} \\
v_F(x, t) = \bar{v}_F(x, t) \quad x \in \Gamma_{vF} \\
p^F(x, t) = \bar{p}(x, t) \quad x \in \Gamma_p
\]

(3.2)

\(\bar{t}^S\) and \(\bar{t}^F\) represent the solid and fluid loading vectors in terms of Neumann boundary conditions. For the Neumann boundary conditions an outward-oriented unit surface normal \(n\) is required to modify the direction. The solid total stress is defined by \(T^S = T^S_E - n^S p^F I\) with the separation of the extra stress term \(T^S_E\) and the stress caused by the pore fluid pressure \(n^S p^F I\). In order to apply surface traction on drained boundaries one can use the relation \(\bar{t} = \bar{t}^S + \bar{t}^F\). It is important to consider that prescribing pressure as the \textit{Dirichlet} boundary conditions for the pore fluid would also result in prescribing the conjugated surface traction \(v_F(x, t)\). Therefore, handling the boundary conditions as they are desired need extra caution. Otherwise, the model would present false results.

Due to the size and complexity of the coupled system of partial differential equations (PDE) an analytical solution is out of the picture for this work. Therefore, the next option would be to use a numerical solution for the set of PDEs. This is done by developing the weak formulation of the strong balance relations.

### 3.1.1 Governing weak formulations of biphasic porous medium

The numerical solution of the set partial differential equations is obtained by the Finite-Element method (FEM). In order to able to use FEM on this set of equations the transformation of the equations into their weak form is required. To do so, one can utilize the method of weighted residuals based on the principal of virtual work (PVW). The main idea behind the method of weighted residuals is to multiply each of the equations by the trial functions and then to integrate the result over the spatial domain \(\Gamma^Q\). This integration would result in energy-like variational statements. The next step would be the utilization of the \textit{Gaussian} divergence theorem and the product rule that would result in the required weak forms of balance equations for the biphasic porous medium. The set of weak formulations are as follows,
3.1 IBVP of the dynamic representation of porous media

\[\int_\Omega \delta u_S \cdot [(u)_S' - v_S] dv = 0,\] (3.3)

\[\int_\Omega \text{grad} \delta u_S \cdot (T^S_E - n^S p^F \mathbf{I}) dv - \int_{\Gamma_{tS}} \delta u_S \cdot \mathbf{t}^S da + \]

\[+ \int_\Omega \delta u_S \cdot \left\{ n^S \rho^S \gamma^F - \frac{(n^F g^F)^2}{k^F} \mathbf{w}_F \right\} dv = 0,\] (3.4)

\[\int_\Omega -\text{div} \delta v_F n^F p^F dv - \int_{\Gamma_{vF}} \delta v_F \cdot \mathbf{t}^F da +\]

\[+ \int_\Omega \delta v_F \cdot \left\{ n^F \rho^F [(v_F)_S'] - \frac{n^F g^F}{k^F} \mathbf{w}_F \right\} dv = 0,\] (3.5)

\[\int_\Omega \delta p^F \text{div} v_S dv - \int_\Omega \text{grad} \delta p^F \cdot n^F \mathbf{w}_F dv + \int_{\Gamma_v} \delta p^F \mathbf{v} da = 0.\] (3.6)

The relation \(v_S = (u_S)_S'\) has been used to reduce the order of the global set of differential equations into first order in time. Furthermore, \(\delta u_S, \delta v_F\) and \(\delta p^F\) are the weight functions, which correspond to the primary variables \(u_S, v_F\) and \(p^F\).

This set of weak formulation was implemented in a PANDAS material routine. As a result, PANDAS was able to solve the initial boundary value problem of the biphasic porous media with specified initial and boundary conditions. Later in Chapter 5, a numerical example of biphasic dynamic porous media representation will be demonstrated.

### 3.1.2 Time discretization of the coupled problem

The mutual momentum exchanges governed by (2.17) causes a strong coupling in the dynamic behavior in the bipasic porous medium. Therefore, neither the set of equations for the solid skeleton nor for the pore fluid can be solved independently. This is the reason that the sets of DE for both subsystems should be solved simultaneously.

To continue, based on the work of Markert et al. [11], in order to integrate large DAE systems, one-step s-stage diagonally implicit Runge-Kutta (DIRK), Butcher [3], methods provide suitable means at moderate storage and computational costs as they allow the solution of the stage equations in a decoupled fashion. Furthermore, for the strongly coupled problems such as the behavior of a biphasic porous medium,
in order to be able to use large time steps, a stiffly accurate method is needed. This would lead to choosing the implicit (backward) Euler (IE) scheme but the problem with this choice is that IE does not react well to the dynamic problems since it has numerical damping. Therefore, TR-BDF2 method which is a combination of 2nd-order backward difference formula and 2nd-order accurate trapezoidal rule (TR), is chosen which does not suffer from the mentioned problems. In table (3.2) TR-BDF2 is demonstrated in the framework of Runge-Kutta (RK) methods and its Butcher table.

\[
\begin{array}{ccccc}
    c_1 & a_{11} & \ldots & a_{1s} \\
    \vdots & \vdots & \ddots & \vdots \\
    c_s & a_{s1} & \ldots & a_{ss} \\
    b_1 & \ldots & b_s \\
\end{array}
\]

Table 3.1: The Butcher tableau for the implicit Runge-Kutta method.

\[
\begin{array}{cccc}
    0 & 0 & 0 & 0 \\
    \theta & \frac{\theta}{2} & \frac{\theta}{2} & 0 \\
    1 & \frac{3\theta-\theta^2-1}{2\theta} & \frac{1-\theta}{2\theta} & \frac{\theta}{2} \\
    & \frac{3\theta-\theta^2-1}{2\theta} & \frac{1-\theta}{2\theta} & \frac{\theta}{2} \\
\end{array}
\]

Table 3.2: TR-BDF2 with $\theta = 2 - \sqrt{2}$.

### 3.2 IBVP of fluid dynamics

In order to solve the IBVP of fluid dynamics, the same approach as the biphasic porous media is pursued with required modifications. As it was mentioned before, in this matter solid constituent $\varphi^S$ will be replaced by a pseudo material which represent the mesh $\varphi^m$. Furthermore, with the help of the MED of the fluid behavior, this constituent (mesh) can move arbitrarily and independent of fluid particles.

As a matter of fact in this approach, as it is shown further in this chapter, the coupling element between two constituents will be eliminated. Therefore, the set of differential equations basically represents two separate problems, which does not
affect each other. These two sets of equations represent the bulk fluid and mesh behavior. Furthermore, the mentioned differential equations have the following elements:

- **Primary variables** \( u(x, t) \): The unknowns of the system of equations under investigation were chosen as

  \[
  u(x, t) = [u_m, v_L, p_L].
  \]  

  Vector \( u(x, t) \) includes the displacement of the mesh \( u_m \), velocity of the bulk fluid \( v_L \) and pressure \( p_L \).

- **Secondary variables**: These are functions of the primary variables that in our case consist of \( v_m \) and \( w_L \), which are respectively mesh velocity and the fluid velocity relative to the mesh displacement.

- **Initial conditions** \( (t = t_0, x \in \Omega) \).

- **Boundary conditions** \( (t \in [0, T], x \in \Gamma) \): The surface \( \Gamma = \partial \Omega \) is split into Dirichlet (essential) and Neumann (natural) boundaries allowing for application of the different external boundary conditions. In particular, these boundaries for given set of equations can be defined in the general form as follows:

  \[
  \begin{align*}
  \text{Dirichlet} & \quad \text{Neumann} \\
  u_M(x, t) &= \bar{u}_M(x, t) \quad x \in \Gamma_{u_M} \\
  v_L(x, t) &= \bar{v}_L(x, t) \quad x \in \Gamma_{v_L} \\
  p^L(x, t) &= \bar{p}(x, t) \quad x \in \Gamma_{p^L} \\
  t^L(x, t) &= \bar{t}^L(x, t) = -p^L \, n \quad x \in \Gamma_{t^L} \\
  v_L(x, t) &= \bar{v}(x, t) = w_L \cdot n \quad x \in \Gamma_v
  \end{align*}
  \]

  As it was discussed before, these sets of boundary conditions are similar to what was taken into account for biphasic porous media, except the changes due to the fact that the pseudo concept of mesh implies that the mesh can not carry any load. Therefore, there is no Neumann boundary condition for the mesh, and also since the whole volume is occupied by the bulk fluid, no volume fraction has been involved in the boundary conditions.

  Apart from the mentioned differences, the other terms that are used here are the same as what was discussed in section 3.1.
3.2.1 Governing weak formulations of the bulk fluid

In order to get the weak counterpart of the balance equations of the bulk fluid, the same procedure of section 3.1.1 is applied. As a result, the weak forms of required balance equations for bulk fluid dynamic representation are as follows:

\[
\int_{\Omega} \delta u_m \cdot [(u_m)_m' - v_m] dv = 0, \quad (3.9)
\]

\[
\int_{\Omega} \text{grad} \delta u_m (T_E^m) dv - \int_{\Gamma} \delta u_m \cdot \bar{t}^m da + \int_{\Omega} \delta u_m \cdot (v_m)_m' dv = 0, \quad (3.10)
\]

\[
\int_{\Omega} -\text{div} \delta v_L p dv - \int_{\Gamma} \delta v_L \cdot \bar{t}^L da + \int_{\Omega} \delta v_L \cdot [p_L (v_L)_m'] dv = 0, \quad (3.11)
\]

\[
\int_{\Omega} \delta p_L \text{div} v_m dv - \int_{\Omega} \text{grad} \delta p_L \cdot (w_L) dv + \int_{\Gamma} \delta p_L \bar{v} da = 0. \quad (3.12)
\]

This set of weak formulation was embedded into a PANDAS material routine for the bulk fluid problem. Later on, when it comes to the coupling of two pandas solvers together in Chapter 4, this physic is used for solving the bulk fluid part in different IBVP.

3.2.2 Numerical treatment of the Navier-Stokes equation

Equation (3.11) is the weak form of the fluid momentum balance for an inviscid fluid, which is derived from (2.37) for \( T_E^k = 0 \). Equation (2.37) is in fact the Navier-Stokes equation to describe the fluid behaviour. The unknowns in this equation are the fluid velocity \( v_L \) and the pressure \( p_L \). One of the main difficulties in a numerical procedure for approximating the solution of the Navier-Stokes equation is introduced by the incompressibility constraint \( \text{div} v_L = 0 \), in which there is no term that couples the pressure \( p_L \) and the velocity \( v_L \). Furthermore, this constraint requires that the so-called Babuska-Brezzi inf-sup condition is fulfilled. There exist a vast literature on numerical approximation of the Stokes equations and the incompressible Navier-Stokes equation. In this work to relax the incompressibility constraint in an appropriate way the so-called “artificial compressibility” method was used as follows:

\[
\text{div} v_L + \beta p_L = 0. \quad (3.13)
\]
In this method which was introduced by Chorin [4] the continuity equation is modified by adding an artificial incompressibility parameter $\beta$ and pressure $p^L$. This method has proven to be efficient in handling velocity-pressure coupling problems in a way that without considering viscous terms, the system of equations becomes hyperbolic and traditional techniques developed for solving subsonic compressible flows can be applied [4].

The parameter $\beta$ can be chosen through a variety of values. In this work $\beta$ was chosen based on recommended values in works of Chorin [4] as follow:

$$\beta \left[ \frac{m^3}{(N \cdot s)} \right] \leq 10^{-7}. \quad (3.14)$$

Furthermore, when the continuity equation has been modified as mentioned, the weak form of it (3.12) which is embedded in PANDAS physic will have the following form

$$\int_{\Omega} \delta p^L \text{div} \, v_m \, dv - \int_{\Omega} \beta \text{grad} \delta p^L \cdot (w_L) \, dv + \int_{\Gamma} \delta p^L \, v \, da = 0. \quad (3.15)$$

### 3.3 Spatial discretization

Transformation of the continuous variational problem into a discrete one is required by the numerical implementation of the FEM. The descritized body $\mathcal{B}$ consists of $N_e$ finite elements and the numerical solution holds for the approximate discrete domain $\Omega^h$. This discrete domain has $N_u$ nodes and all the weak formulaion for the set of PDE are solved on each node of the dicrete domain. This process results in having the complete sets of PDE for the whole nodes on the whole domain as:

$$\mathbf{u}^h(x, t) = \bar{\mathbf{u}}^h(x, t) + \sum_{i=1}^{n} N_{u(i)}(x) \, \mathbf{u}_{(i)}(t) \in \mathcal{S}^h_u(t),$$

$$\delta \mathbf{u}^h(x) = \sum_{i=1}^{M_u} M_{u(i)}(x) \, \delta \mathbf{u}_{(i)} \in \mathcal{T}^h_u. \quad (3.16)$$

In order to be able to solve the weak form of the balance relations the Dirichlet boundary conditions are approximated and represented in vector $\bar{\mathbf{u}}^h$ containing the Dirichlet BCs for the nodes on the boundary. All the unknowns are gathered in the unknown filed $\mathbf{u}$ and $N_u$ is the number of FE nodes used for the approximation of the unknowns at each node of the FE mesh and $N_{u(i)}$ denotes the space-dependent nodal coefficient. Similarly, with $M_u$ as the number of FE nodes used for the test functions in $\delta \mathbf{u}$, $M_{u(i)}$ are the elements of the matrix with global basis functions and $\delta \mathbf{u}_{(i)}$ is the vector containing the values for the nodal test functions. Furthermore,
based on the Bubnov – Galerkin procedure same functions are utilized for \( \mathbf{M}_{u(i)} \) and \( \mathbf{N}_{u(i)} \) for the approximation of unknown filed and test functions. In this case, the trial and test spaces coincide except for a shift through the Dirichlet boundary conditions, i.e., \( \mathcal{S}_u^h(t) = \bar{u}^h + \mathcal{T}_u^h \).

Furthermore, the following trial functions were defined:

\[
\mathbf{u}_\alpha \approx \mathbf{u}_\alpha^h = \mathbf{U}_\alpha \mathbf{u}^{nd}_\alpha.
\]  

(3.17)

Therein, \( \mathbf{U}_\alpha \) contain the interpolation function corresponding to the nodal unknowns \( \mathbf{u}_\alpha \) of the FE mesh. Spatial discretization of the sub-domains could be done following standard FE procedures, using linear or higher-order interpolation functions.

Moreover, the semi-discrete FE-Galerkin formulation of the variational problem in the course of the FE discretization with linear elastic solid behavior reads:

Find \( \mathbf{u}^h \in \mathcal{S}_u^h(t) \) such that

\[
\mathcal{G}_u^h(\delta \mathbf{u}^h, \mathbf{u}^h) = 0 \quad \forall \delta \mathbf{u}^h \in \mathcal{T}_u^h, \quad t \in [t_0, T].
\]

(3.18)

Where \( \mathcal{G}_u \) is a function vector including the weak formulations.
Chapter 4: Fluid-Porous-Media Coupling

The numerical simulation of fluid-porous-media interaction phenomena arises in many fields of engineering including biomechanics, soil mechanics, and a large class of aerospace engineering problems. Several approaches have been proposed in the past for solving FPMI problems on moving and deforming meshes, among which, as it will be presented in this chapter the Conventional Serial Staggered (CSS) algorithm has been used in this work.

If \( \mathbf{v}_L \) denotes the velocity of the bulk fluid in the MED and \( p^L \) its pressure and \( \mathbf{v}_P \) represent the velocity of the biphasic porous medium (including both constituents) field, \( \sigma_P \) and \( \sigma_L \) the porous material stress tensor and the fluid stress tensor, \( \Gamma \) the fluid-porous-media interface boundary (wet boundary of the porous material) and \( \mathbf{n} \) the normal at a point to \( \Gamma \), then the FPMI equations can imposed as

\[
\sigma_P \cdot \mathbf{n} = -p^L \cdot \mathbf{n} + \sigma_L \cdot \mathbf{n} \quad \text{on} \quad \Gamma, \tag{4.1}
\]

\[
\mathbf{v}_P = \mathbf{v}_L \quad \text{on} \quad \Gamma.
\]

The first of these two interface boundary conditions states that the tractions on the wet surface of the porous material are in equilibrium with those on the bulk fluid side of \( \Gamma \). The second equation expresses the compatibility between the velocities of the porous material and the bulk fluid at the fluid-porous-medium interface. For inviscid flows, the second equations is replaced by the slip wall boundary condition

\[
\frac{\partial \mathbf{u}_P}{\partial t} \cdot \mathbf{n} = \frac{\partial \mathbf{u}_L}{\partial t} \cdot \mathbf{n} \quad \text{on} \quad \Gamma, \tag{4.2}
\]

Furthermore, the porous material and dynamic mesh motions are also coupled by the equation (4.3) to fulfill the desire to attach the mesh to the moving boundaries.

\[
\frac{\partial \mathbf{u}_m}{\partial t} = \frac{\partial \mathbf{u}_P}{\partial t} \quad \text{on} \quad \Gamma, \tag{4.3}
\]

The main objective of this chapter is to overview the solution by partitioned procedure of the discrete representative IBVPs through a staggered algorithm.
4.1 Partitioned solution procedure

In case of geometrically simple and small-strains FPMI problems, a "monolithic" fully explicit or fully implicit treatment of the coupled FPMI equations of motion is possible. On the other hand, for more complex problems, each of the four components of the coupled problem including the solid skeleton and pore fluid in the porous medium and the bulk fluid and mesh has different mathematical and numerical properties, well established but distinct numerical solvers, and readily available commercial softwares. For the fluid equations are nonlinear, the porous material equations and the semi-discrete equations governing the pseudo-structural fluid grid system may be linear or nonlinear. The matrices resulting from a linearization procedure are in general symmetric for the structural problem, but they are typically asymmetric for the fluid problem. Consequently, the simultaneous solution of all equations by a monolithic scheme is in general computationally challenging, mathematically and economically suboptimal, and software-wise unmanageable. Alternatively, these can be solved by a partitioned or staggered procedure, cf. [10]. Each partition will be discretized spatially and temporally according to the preferred schemes, and for both partitions PANDAS will be the main solver. An elementary yet popular partitioned scheme for solving the FPMI coupled problem is as follow:

1. solve the bulk fluid with external boundary conditions and assumed boundary conditions on the wet boundary,
2. accordingly, update the fluid dynamic mesh and transfer the velocity to the porous medium,
3. solve the porous partition and get the new pore fluid pressure and solid stress field,
4. transfer the updated pressure and stress fields into pressure for the bulk fluid,
5. solve the bulk fluid under the updated loading.

In case of need for more accurate procedure, predictor/corrector iterations can be added within each cycle of the five-steps staggered scheme, which make the overall procedure a strongly coupled solution algorithm.

4.2 The conventional serial staggered algorithm

The basic staggered algorithm mentioned above is referred to as the Conventional Serial Staggered (CSS) procedure. CSS is graphically depicted in Figure 4.1 wherein, W and U denote the porous medium and bulk fluid state vector, p denotes the fluid
4.2 The conventional serial staggered algorithm

pressure, and the subscription \( n \) refer to the \( n^{th} \) time step. The CSS method has the highest popularity among partitioned procedures because of its simplicity.

In case that, the fluid flow requires a finer temporal resolution than the porous medium, the coupling time step \( \Delta t \) will be typically dictated by the time step \( \Delta t_F \) that guarantees the specific precision in the fluid solution, rather than the time step \( \Delta t_P > \Delta t_F \) that satisfies the required accuracy for porous medium. The minor implementation advantages of using the same time step for both fluid and porous media solvers can not outweigh the substantial computational advantages of sub-cycling the fluid computations with a factor \( n_{P/F} = \Delta t_P / \Delta t_F \). The main advantage of this procedure is the saving in the overall simulation CPU time because in that case, CPU cost of solving the porous medium will be less.
The CSS method equipped with fluid sub-cycling is demonstrated in Figure 4.2.

Furthermore, as demonstrated in [14], the CSS procedure has the disadvantage that it is only first-order time-accurate, regardless of using solvers with higher accuracy for each sub-system. Therefore, references such as [14] recommend fully implicit monolithic solution procedures.

4.3 Coupling two PANDAS solvers

The main interest of this work is to use separate PANDAS solvers for each of sub-systems in the coupling process. The main motivation for this approach instead of using an external fluid solver, e.g., ELMER, is the computational advantages that using the same environment for both porous medium and the bulk fluid provides. Furthermore, alongside the process of solving the FPMI problem, the developed physic for fluid can be used for further contributions in any problem related to Navier – Stokes governed fluid phenomena in future works. The coupling process of two separate PANDAS solvers in a staggered perspective will be described in following sections.
4.3.1 Implementation of CSS for two PANDAS solvers

The underlying algorithm of coupling two PANDAS solvers for this project is the same as the CSS. The main idea behind this procedure consists of using two separate PANDAS implementations of fluid and porous media, as what was described previously in this chapter in an staggered way. This purpose is fulfilled by running the code for each subsystem for one time step and transferring required information between solvers for next time step and repeating the process as the time goes on. Herein, the main elements of the coupling process are listed:

- **Batch command**: this code controls the master procedure. It starts and halts each separate PANDAS solver at each time step and it also runs the required python code for transferring required data between two solvers.

- **Manager**: a python code that reads, writes, and translates the data for the sake of communication between two solvers. In order to be able to use this python code, some modifications in the core of PANDAS was required.

- **B.C. file**: The boundary condition file is where output of each solver at each time step is being written on and also being read from. Manager translate and writes data on the B.C. file and also reads from it and according to the data written on B.C. file, manager updates the boundary conditions for the solvers in next time step.

- **Porous Media Solver**: the routine for porous media in PANDAS, which its boundary conditions gets updated at each time step and solves the new problem for the new boundary conditions at each time step.

- **Fluid Solver**: the routine for fluid in PANDAS which its boundary conditions gets updated at each time step and solves the new fluid flow for the new boundary condition at each time step.

The process of staggered solution of FPMI problem is demonstrated in Figure 4.3
In this flowchart at each vertical section including the initiation of the process and steps number 1 and 2 PANDAS solves one IBVP of one of the subsystems and then move to the next vertical section. The initiation takes place only at the beginning of the process and then a loop of steps number 1 and 2 would proceed through the desired time for solving the problem.

### 4.3.2 Communication between the solvers

The main idea behind the communication between the solvers according to the CSS procedure is updating boundary conditions and geometry for each subsystem at each time step. This is be done by means of two elements: the Manager and the B.C. file. Therein, at each time step, Manager reads the output data of the solved subsystem and gathers the required data from the output. Then it writes the required data on the B.C. file in a translatable manner for PANDAS. Furthermore, before solving the other subsystem, the new information from B.C. file will be translated as the new boundary condition and geometry and the solver proceeds and this cycle continues.
4.4 Detailed algorithm

until the end of time span. this process is demonstrated in Figure 4.4.

![Figure 4.4: Updating boundary conditions and geometry](image_url)

The flow of data is demonstrated in Figure 4.4. The process takes place either at right side or left side of the flowchart at a time. Each solver constantly transfers data to its output and based on the output the B.C. file gets updated and the process shifts to the other solver.

### 4.4 Detailed algorithm

Proceeding form the overview of the CSS implementation in this work, herein, the process is explained in details.

The CSS algorithm detailed implementation:

1. Batch command runs PANDAS for bulk fluid with initial conditions for one timestep. On the wet boundary the initial value for pressure will be set to $p^L \equiv 0$.

2. Manager initiate.

   (a) Manager reads .dat output file produced by PANDAS from previous step.

   (b) Manager finds the nodes on the wet boundary.
3. B.C. file will be initiated.

(a) $u_m$ and $v_L$ are written in B.C. file.

(b) Batch command starts PANDAS for porous medium for one time step.

i. Boundary conditions and geometry for porous medium will be updated based on information of B.C. file, which means on the wet boundary, the velocity of both solid skeleton $v_S$ and the pore fluid $v_F$ are equal to the velocity of the bulk fluid $v_L$ that has been calculated in previous step.

ii. PANDAS solves biphasic porous media subsystem.

(c) Manager reads the displacement of the solid skeleton $u_S$ and the total stress of the solid skeleton $T^S$ from the output of PANDAS for porous medium.

(d) $u_S$ and $T^S$ are written in B.C. file.

4. Batch command starts PANDAS for fluid.

(a) Boundary conditions and geometry are updated for fluid as displacement of the mesh and pressure of the bulk fluid for the nodes on the interface.

(b) PANDAS solves bulk fluid as the new IBVP.

5. The process starts again from the third step.

### 4.5 Translation of the output to the boundary conditions

Translating the output of each time step for each subsystem into boundary conditions and geometry for the next time step for the next subsystem is the most crucial part of the CSS implementation in a fluid-porous-media interaction problem in PANDAS. To do so, as it was explained in previous section, required information
4.5 Translation of the output to the boundary conditions

will be gathered from results of each subsystem for each time step and then these
results are translated into boundary conditions and geometry embracing the other
subsystem.

By solving IBVP for the bulk fluid in PANDAS environment, a set of information is
produced as output. From this set, fluid velocity of the nodes on the wet boundary
are collected. This process is done considering the condition that for the first time
step it is assumed that pressure on the wet boundary is equal to zero. Therefore,

\[ p \equiv 0 \text{ on the we boundary for first step,} \]

\[ U_L = \{u_m, v_L, p^L\}. \]  

Following this, displacement of the mesh is directly side considered as the *Dirichlet*
boundary condition for the solid skeleton on the porous media.

\[ u_m \equiv u_S. \]  

Moreover, the velocity of fluid is considered as velocity of solid skeleton \( v_S \) as well
as the pore fluid \( v_F \) on the wet boundary. This will result in zero seepage velocity
\( w_F \) on the boundary.

\[ v_L \equiv v_S, \]

\[ v_L \equiv v_F, \]

\[ \rightarrow w_F = 0. \]  

Therein, by performing the same procedure for IBVP of the porous medium, the
required set of information from the output data includes the fluid velocity, the fluid
pressure, and the displacement of the solid skeleton.

\[ U_P = \{v_F, p^F, u_S\}. \]  

Furthermore, to translate the outcome of the porous media IBVP into boundary
conditions to solve the IBVP of the bulk fluid, applying pressure on the bulk fluid is
calculated based on the resulting total stresses solid skeleton for the porous material.

\[ p^L = t^S \cdot n \]

\[ w_L = v_L - v_m = 0. \]  

The important part is that \( w_L = 0 \) results in the coherence of the movement of the
mesh and the bulk fluid on the wet boundary.
Applying this algorithm to exchange and update information for each IBVP (bulk fluid and porous medium) results in CSS procedure for the coupled problem. Numerical examples of this procedure are presented in Chapter 5.
Chapter 5: Numerical Examples

In this chapter, numerical examples for dynamic biphasic porous media, dynamic fluid flow, and fluid-porous-media interaction are represented. These numerical calculations have taken place in PANDAS environment using the governing equations explained in Chapter 3. In case of FPMI, staggered solution explained in Chapter 4 was utilized to solve the problem. Details of each example and the results are as follows.

5.1 Dynamic representation of biphasic porous material

In this example, the biphasic porous subsystem is represented under cyclic loading. The geometry of the model is shown in Figure 5.1.

Figure 5.1: Geometry and spatial discretization for the biphasic example.

As it is demonstrated in Figure 5.1, the first example is a 2-d, 2m * 5m rectangle-shaped porous medium. It is spatially discretized by linear modified Taylor-Hood finite elements. The boundaries are four walls which are name I to IV. The material properties of the model are as follows
Material Parameter | Symbol | Value | SI unit
--- | --- | --- | ---
1st Lamé constant of solid skeleton | $\mu^S$ | $5.583 \cdot 10^6$ | N/m²
2nd Lamé constant of solid skeleton | $\lambda^S$ | $8.375 \cdot 10^6$ | N/m²
Effective density of solid | $\rho^{SR}$ | 2000 | kg/m³
Effective density of pore fluid | $\rho^{FR}$ | 1000 | kg/m³
Initial volume fraction of solid | $n_{0S}$ | 0.67 | -
Darcy permeability | $k^F$ | $10^{-5}$ | m/s

**Table 5.1:** Material parameters of the biphasic porous medium.

Furthermore, boundary conditions and time-dependent loading pattern of dynamic compression of the biphasic saturated column is demonstrated in **Figure 5.2**. A cyclic loading pattern is chosen with fully drained boundary conditions for pressure to see the pore fluid flow in and out of the porous medium.

$$\bar{t}_y = 1000 [1 - \cos(20\pi t)] N$$

**Figure 5.2:** Boundary conditions and loading path of dynamic compression of the biphasic saturated column

The results of this simulation is presented in following diagrams. In **Figure 5.3** the solid skeleton displacement of the top part of the column is presented and in **Figure 5.4** the seepage velocity at top of the column is shown.
5.2 Dynamic fluid flow

As it was expected, a cyclic dynamic behavior is presented here for the biphasic porous media. This figurative benchmark confirms validity of this model for the dynamic behavior of biphasic porous medium. One can see that the numerical solution of the problem matches the physical expectations in a way that in response to the cyclic loading, the pore fluid goes in and out of the porous material alongside the compression of the whole porous medium. In the next step, the model for bulk fluid will be used to represent two academic examples of dynamic flows.

5.2 Dynamic fluid flow

In this section two examples of numerical solution for dynamic flow in PANDAS environment are presented. Governing equations discussed in Chapter 3 are taken into account to model these problems. The first numerical simulation demonstrates
an academic problem of a viscous incompressible flow, and the second one models the dynamic inviscid incompressible flow in a pipe with moving boundaries. It is worth to mention that MED representation was used in both examples. It enables the model to handle moving boundaries in case it is needed.

5.2.1 Dynamic flow of viscous incompressible fluid

Simple flow of a viscous fluid in a pipe has the dynamic velocity profile through its flow section, which can be a satisfying benchmark for the simple IBVP of the fluid dynamic. Therefore, the model with the geometry similar to the previous example and spatial discretization as shown in Figure 5.5 is developed. One main difference in case of spatial discretization is the number of elements which are used in horizontal direction. The reason behind this choice lies beneath the fact that the selected finite element withholds linear shape functions; therefore, in case of using one element in horizontal direction the FEM model will not be able to represent the physical fluid velocity profile in horizontal direction. Furthermore, increasing the number of elements will accordingly result in a more precise result.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure5.5.png}
\caption{Geometry and spatial discretization of the viscous fluid flow model}
\end{figure}

Similar to the previous example, modified Taylor-Hood elements are utilized for the spatial discretization. Moreover, the material parameters are presented in Table 5.2. In this example, the viscosity of blood was taken into account as our viscous fluid, Rand et al. [15].

The boundary conditions of the dynamic fluid flow is demonstrated in Figure 5.6. A constant pressure gradient $\Delta p$ is applied to two ends of the column in order to examine the characteristics of the viscous flow representation of the model with the expected fluid velocity profile in the $x$-direction.
5.2 Dynamic fluid flow

<table>
<thead>
<tr>
<th>Material Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>SI unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st Lamé constant of elastic mesh</td>
<td>$\mu^m$</td>
<td>$5.583 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>2nd Lamé constant of elastic mesh</td>
<td>$\lambda^m$</td>
<td>$8.375 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>Effective density of mesh</td>
<td>$\rho^{mR}$</td>
<td>1</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Effective density of bulk fluid</td>
<td>$\rho^{LR}$</td>
<td>1000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Fluid dynamic viscosity</td>
<td>$\mu^L$</td>
<td>$3.5 \cdot 10^{-4}$</td>
<td>kg/(s·m)</td>
</tr>
</tbody>
</table>

Table 5.2: Material parameters of the viscous fluid.

The resulting fluid velocity through $x$-axis is represented in Figure 5.7 and Figure 5.8. These two velocity profiles are demonstrated at $t = 2$ and $t = 5$. Due to the constant pressure gradient in the $y$-direction, linear increase in the velocity gradient in $x$ direction ($\partial_x \mathbf{v}$) was expected. As it is shown in the diagrams for the profiles of the fluid velocity in the $y$-direction, on the walls of the pipe the velocity is equal to zero and by moving moving toward the middle of the pipe the value of the velocity in the $y$-direction tends to increase. This aspect of the results means that the profile of the velocity becomes sharper as the time goes by. It is confirmed by the fact that the pressure gradient is constant through the time so the fluid velocity would increase through the time in the middle of the pipe.
Furthermore, the fluid velocity profile over the whole column is presented in Figure 5.9. Herein, since there is a constant pressure gradient $\partial_y p = \text{const.}$, the constant velocity profile through the column confirms the validity of the model. Moreover, due to no-slip condition on the lateral walls, the fluid velocity remains zero throughout the time on the walls.

\[ \begin{align*}
\text{Figure 5.7: Fluid velocity profile through } x \text{ direction at } t = 2 \text{ S} \\
\text{Figure 5.8: Fluid velocity profile through } x \text{ direction at } t = 5 \text{ S}
\end{align*} \]
Figure 5.9: Fluid velocity profile through the whole column at $t = 2\text{S}$. The color range in the image starts from red= 0 and gets to blue> 0.

5.2.2 Inviscid fluid flow model with moving boundaries

As it was stated in Chapter 2, the main purpose of using the MED for fluid representation is the ability to demonstrate fluid behavior in IBVP with moving boundaries. In most cases, the boundary conditions and geometry of the fluid subsystem in FPMI problems are actively changing. Therefore, the fluid model should be able to represent a faultless behavior of the fluid flow with moving geometry and boundary conditions.

In order to put the reaction of the fluid model to this phenomenon to test, a column of fluid with same geometry as previous example is being tested when one of the lateral walls moving as a function of time. The material properties of the fluid are exactly the same, except the fact that in this representation, an inviscid incompressible fluid is being modeled. Furthermore, the boundary conditions are shown in Figure 5.10.

In this problem, fluid can go inside and outside through wall number I and other three walls are impermeable. Due to the cyclic motion of the lateral wall, a cyclic asymmetric flow of fluid in both $x$- and $y$-directions is expected.

The results of this simulation is demonstrated in Figure 5.11 and Figure 5.12, regarding the distribution of the fluid velocity in the pipe for two different time steps in each phase of the periodic motion of the lateral wall.
Chapter 5: Numerical Examples

Figure 5.10: Boundary conditions and moving wall.

Figure 5.11: Fluid velocity in the $y$-direction when the wall is moving toward left at $t = 1S$. The color range presented in the image starts from blue < 0 to red > 0.

The model represents a periodic behavior under the periodic motion of the walls. Moreover, as the wall moves towards the center, the fluid is pushed out of the pipe with the velocity profile demonstrated in Figure 5.11 and when the wall moves in the other direction, the fluid finds the exact opposite velocity profile and the water it is sucked in the pipe which is the opposite of being pushed out of the pipe. This behavior matches the expectations of an inviscid incompressible fluid. Therefore, with these two academic benchmarks represented in last two examples, the fluid model satisfies the necessary requisites to act as the fluid subsystem in the FPMI.
Figure 5.12: Fluid velocity in the $y$-direction when the wall is moving toward right at $t = 1.2xS$. The color range presented in the image starts from blue<0 to red>0.

In the next section, a FPMI problem will be solved by means of the presented porous medium model, fluid model, and the CSS algorithm for the coupling process.
5.3 Numerical example of the FPMI problem

As the final purpose of this work, a 2-d fluid-porous-medium interaction problem is presented. The Conventional Serial Staggered algorithm (Chapter 4) with different values for time step is used to solve this problem. Furthermore, the results of this example are qualitatively and quantitatively analyzed. This example explores the interaction between two rectangular sub-domains with geometry and boundary conditions represented in Figure 5.13 and material properties obtained form Table 5.3.

Figure 5.13: Geometry and boundary conditions of the numerical FPMI example.

BC for the bulk fluid

II, IV, VI, VIII:
\[ v_{Mx} = 0 \quad v_{Lx} = 0 \]

III:
\[ v_{Mx} = 0 \quad v_{Lx} = 0 \quad p^L = 100 \text{ N/m}^2 \]

V:
\[ v_{My} = 0 \quad v_{Ly} = 0 \]

VII:
\[ v_{Mx} = 0 \quad v_{Lx} = 0 \quad p^L = 105 \text{ N/m}^2 \]

BC for the porous medium

IX, XIII:
\[ v_{Sx} = 0 \quad v_{Fx} = 0 \]

X, XII:
\[ v_{Sx} = 0 \quad p^F = 0 \text{ N/m}^2 \]

XI:
\[ v_{Sx} = 0 \quad v_{Fx} = 0 \]
5.3 Numerical example of the FPMI problem

Because of the conditional-stability in the nature of the CSS algorithm, the time-step plays a significant role in the stability of the solution. This conditional-stability lies beneath the mathematical nature of the CSS algorithm, cf. [10]. As explained in the CSS coupling algorithm (Chapter 4), the initiation of the problem starts with solving the fluid sub-domain without considering the porous medium. Although this step is necessary, it contributes to conditional-stability of the solution with high level of importance. This contribution is a result of neglecting the presence of the porous medium in the first time step for the bulk fluid. This neglecting results in a major error in coupling process which is the very big displacement applied on the porous medium in the first time step. The resulting displacement has a considerable error, because at the time of calculation in the bulk fluid the coupling was not considered. Therefore, the applied displacement is too big. This error is highly dependent on the selected value for the time-step size and in case of proper value of time-step size the procedure will converge to the expected coupling behavior. This phenomenon will be further explained in the next sections.

The spatial discretization of the problem was chosen based on the geometry and boundary conditions of the problem. For each sub-domain, a regular rectangular 1.0 m × 1.0 m mesh was chosen which can be considered a fine mesh for the properties

<table>
<thead>
<tr>
<th>Material Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>SI unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st Lamé constant of elastic mesh</td>
<td>$\mu^m$</td>
<td>$5.583 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>2nd Lamé constant of elastic mesh</td>
<td>$\lambda^m$</td>
<td>$8.375 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>Effective density of mesh</td>
<td>$\rho^{mR}$</td>
<td>1</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Effective density of bulk fluid</td>
<td>$\rho^L$</td>
<td>1000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Bulk fluid dynamic viscosity</td>
<td>$\mu^L$</td>
<td>0</td>
<td>kg/(s·m)</td>
</tr>
<tr>
<td>1st Lamé constant of solid skeleton</td>
<td>$\mu^S$</td>
<td>$5.583 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>2nd Lamé constant of solid skeleton</td>
<td>$\lambda^S$</td>
<td>$8.375 \cdot 10^6$</td>
<td>N/m²</td>
</tr>
<tr>
<td>Effective density of solid</td>
<td>$\rho^{SR}$</td>
<td>2000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Effective density of pore fluid</td>
<td>$\rho^{FR}$</td>
<td>1000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Initial volume fraction of solid</td>
<td>$n_{i0S}$</td>
<td>0.67</td>
<td>-</td>
</tr>
<tr>
<td>Darcy permeability</td>
<td>$k^F$</td>
<td>$10^{-2}$</td>
<td>m/s</td>
</tr>
</tbody>
</table>

Table 5.3: Material parameters of fluid-porous-media interaction problem.
of this specific problem. The same kind of elements as before were used for both sub-domains.

Considering the geometry and boundary conditions of the IBVP, the expected behavior of the problem is as follow:

- With the initiation of the problem, since both sub-domains are at rest at the interface, there is not a considerable force from the porous medium toward the bulk fluid. This results in fluid flow in the direction of porous medium. As a result of incompressible fluid flow toward porous medium, its solid skeleton and pore fluid will have a vertical displacement at the interface and because of draining boundary conditions at the side-bottom of porous medium the horizontal fluid flow at side-bottoms of porous medium are expected.

- As the times goes by, because of the displacement of the porous medium at the interface and the fact that solid skeleton is bounded in porous medium at other boundaries, the stress in solid skeleton and, therefore, the pressure in the bulk fluid arise. This increase in stress will be transferred to bulk fluid and will cause the decrease of fluid flow in bulk fluid in the vertical direction.

- After some point, the increasing pressure of the bulk fluid on the interface will result in cease of fluid flow in vertical directions and as a result of horizontal pressure gradient in bulk fluid, the fluid flow will become horizontal.

These three main stages of the selected IBVP are demonstrated in Figure 5.14. Through snapshots of the simulation of the IBVP, three different stages were chosen with distinct behavior for pore or bulk fluid velocity in each subsystem as a measure for what was expected to happen in this particular coupling problem. These different stages of the represented FPMI are shown in three different slices of the whole process. The fluid and the porous medium subsystems are shown in the upper and lower of the Figure 5.14. The bulk and the pore fluid velocities are shown by vectors in these three snapshots of the process and the process in which the changes of the direction and the qualitative demonstration of the values of these velocities from the simulation in both subsystem take place match the expectation of these coupling problem. Through the different stages of the coupling process as it is demonstrated in Figure 5.14 initiation, dynamic formation, and the final stable stage take place.
5.3 Numerical example of the FPMI problem

Figure 5.14: Snapshots of the behavior of each of the subsystems in selected times as $t = 0.01\,\text{S}$, $t = 0.1\,\text{S}$, and $t = 1.0\,\text{S}$ . The blue vectors show the bulk and pore fluid velocities. Pressure is represented by the color range in the background.

A closer look at the interface of two subsystems provides quite a number of useful and interesting information of our specific FPMI problem. To do so, bulk fluid velocity at the interface and also total stress from the porous medium for two points (A and B) of the interface are shown in the diagrams in Figure 5.15 and Figure 5.16. The location of these points are specified in Figure 5.13. The two diagrams in Figure 5.15 show detailed results of the simulation through 1.5 seconds of the process. The three main stages of the FPMI problem solution with the CSS algorithm is also accented in these diagrams. The first stage, which is the initial part (with relatively enormous oscillation), is shown in the smaller diagrams inside each big one (due to the differences in the scales with the later results).
This oscillation is mainly the result of two different numerical and physical facts. First of all, as it is stated before, the way CSS algorithm initiates the coupling of two subsystems will definitely result in oscillatory response with the assumption of zero pressure for initial condition on the wet boundary since it will result in non-physical displacement, but the process would converge toward the physically expected results shortly afterward. Moreover, a saturated porous material would respond in the same way as a viscoelastic material under loading. This viscoelastic-type response is also likely to contribute in the oscillation observed in the first 0.2 seconds of the simulation, which will result in the types of oscillation shown in the smaller diagrams. Further on, the whole process tends to converge to the stable manner in which the vertical velocity of the bulk fluid (and therefore, the vertical velocity of the porous material and pore fluid) on the wet boundary tends to zero which is an expected phenomenon.

Another interesting aspect of the results demonstrated in these diagrams is the presence of oscillations even after the initial stage. This oscillation is purely due to the nature of CSS algorithm, and has its roots in the numerical scheme of use. One can observe that as the time goes by, the oscillations tend to get smaller although as it is stated before they will never go away. Moreover, the size of this oscillation directly depends on the size of the time step and naturally with smaller time-step sizes the oscillation would become smaller.

In the next set of diagrams presented in Figure 5.16, the total stress from the porous material (and respectively the pressure of the bulk fluid) on the wet boundary at points A and B is presented. Some similar phenomena in addition to some new features of the FPMI problem can be explored in this set of detailed results. The same initial relatively big oscillation occurs from the beginning of the simulation, which is presented in the smaller diagrams inside each figure. Once again, the reason behind this oscillation are mainly the way the CSS handles the coupling problem in the stage of initiation and also the nature of coupling process utilized by the CSS
algorithm. By proceeding with the simulation, the pressure also converges toward its stable value, and after 1.5 seconds there exists almost a constant pressure for each point on the wet boundary, which once again matches the expectations.

One other interesting aspect of the FPMI problem that can be seen in this set of diagrams is the difference between the pressures at points A and B for their stable stages. Due to the fact that point B sits closer to the wall with the higher pressure, it has a higher final pressure than the point A which is located in the middle. This fact can be explored further by paying attention to the complete horizontal bulk fluid velocity which is constant through the most parts of fluid subsystem, which is the result of a constant horizontal pressure gradient. The linear distribution of the pressure from left to right in the fluid subsystem also can be considered as another way that model matches the expectations for its stable final solution.

Further on, in the diagram of Figure 5.17, the results of the CSS method is compared to the results of the LLM method, cf. Chapter 1, for the exact same problem.

The FPMI problem can be solved by means of numerous decoupling solutions from which LLM method provides a similar in basis and somewhat different approach toward the problem, Park et al. [13]. The main difference between these two partitioned schemes is the fact that in the LLM method another subsystem would be defined in the process of the numerical process as the interface and on this subsystem local Lagrange multipliers would be calculated and considered as coupling factor between two subsystems. This approach makes the LLM method unconditionally stable which is a big advantage over the CSS but it will compensate in terms of complexity and computational cost. In case of this exact problem, the time needed for the LLM method to solve the problem for 1 seconds of simulation was almost 10 times bigger than the time needed by CSS algorithm which can be a drawback if a bigger problem with finer mesh is to be solved.

Nevertheless, by solving the problem with the LLM model developed by S. Zinatbakhsh and D. Koch at Institute of Applied Mechanics, University of Stuttgart, a
Figure 5.17: Comparing the results of the CSS method with the results of the LLM method.

A benchmark for the results of this work is demonstrated. In Figure 5.17 the results of the LLM method are shown in blue color for the vertical velocity of bulk fluid in the middle of the wet boundary and the results from the CSS algorithm are shown by black color. As it is shown, the overall behavior and also the ending results match for both algorithms in a way that after 1 second of simulation, the difference between the results of two algorithms is less than $10^{-6}$.

Figure 5.18 demonstrates the deformation of the mesh in the porous medium and bulk fluid respectively. One thing that needs to be taken into account is the fact that these deformation has been enormously magnified in order to be seen in this image. The mesh in the bulk fluid is stretched due to the fact that on the boundaries it is attached to the porous medium and the mesh inside the porous medium is compressed during the simulation. This deformation of the mesh is possible since the MED was used for the bulk or pore fluid representations in both subsystems.

In the next section, the conditional stability of the CSS algorithm will be explored by running the simulation with different time-step sizes.

### 5.3.1 Conditional stability of CSS

As it has been stated in Chapter 4, the CSS algorithm is a conditionally stable solution scheme. So in order to have a converging result, the time should be chosen with caution. The exact critical $\Delta t$ can be found through a stability analysis pro-
5.3 Numerical example of the FPMI problem

As it is demonstrated in Figure 5.19, when computing the pressure on the wet boundary for the bulk fluid in the middle of the interface by having a time step bigger than the critical time-step size, after 0.05 seconds the results start to diverge (shown in red color) which is not the case for smaller chosen time-step sizes.

Further on, by comparing the results of $\Delta t = 10^{-3}$S and $\Delta t = 10^{-4}$S no meaningful difference can be spotted therefore the bigger time-step can be chosen safely.

Going over the results presented in this chapter for the FPMI problem shows that the implemented CSS algorithm in PANDAS environment assisted by the developed side-codes can be used for different FPMI problems. Keeping in mind the limitations of this method, it can also be used to different problems with different scales. With the required modifications, it can also be used wherever coupling of two or more physically different environments is needed. In the next chapter, an overview of
what has been done during this master thesis in addition to outlooks of the work will be presented.

Figure 5.19: results of the model with $\Delta t = 10^{-2}\text{S}, 10^{-3}$, and $10^{-4}\text{S}$
Figure 5.20: Comparison of the results for $\Delta t = 10^{-3} S$ and $\Delta t = 10^{-4} S$. 
Chapter 6: Outlook

Throughout this master thesis a staggered solution of the fluid-porous-media interaction in PANDAS environment was developed. In order to develop a dynamic model for a biphasic porous media, in Chapter 2 the fundamentals of the Theory of Porous Media (TPM) was discussed. With the help of the TPM, the basic formulation for the equations governing the dynamic behavior of a biphasic porous media was developed. These governing equations are based on momentum balance relations of pore fluid and solid skeleton, mixture volume balance and order reducing equation for the velocity and displacement. Furthermore, demanded constitutive equations regarding the pore fluid and solid skeleton were presented in case of a linear elastic solid and inviscid incompressible pore fluid. In Chapter 2, modified Eulerean description (MED) for the bulk fluid subsystem of the FPMI problem was explored with a comparison between different fluid representation methods including Lagrangean, Eulerian, and MED. Navier – Stokes equation regarding the bulk fluid behavior and constitutive equations in case of viscous and inviscid fluid are the last subjects of Chapter 2.

Moreover, in Chapter 3 the process of numerical solution for biphasic porous media and dynamic fluid flow initial-boundary-value problems are discussed. Detailed properties of boundary conditions applied on both porous medium and bulk fluid subsystems, required to establish a numerical solution based on Finite Element Method for spatial discretization of the IBVP are presented in Chapter 3. Furthermore, weak formulations of the balance equations, necessary for implementation of IBVP in PANDAS environment in addition to time discretization methods utilized such as RK methods are described in this chapter.

In order to prepare two separate models for bulk fluid and porous medium, the crucial part in the process of developing a numerical solution for the fluid-porous-media iteration problem is the coupling algorithm. In Chapter 4 the staggered solution and partitioning procedure for solving the FPMI problem is presented. In this work, the conventional serial staggered (CSS) algorithm is chosen due to its computational benefits alongside its simplicity in implementation. The elements needed to execute CSS coupling algorithm in PANDAS environment and the detailed process of coupling in an serial staggered way are the main aspects of Chapter 4. Moreover, in order to perform the CSS algorithm for the FPMI problem, a translation between the results of each subsystem and boundary conditions of the the other subsystem for the next time-step is required. The idea behind this translation and justification of chosen means are provided in Chapter 4.
Finally, in Chapter 5 numerical examples for the developed models are presented. Starting with the dynamic behavior of a biphasic porous media under periodic loading, with prescribed initial and boundary conditions, the model is tested and the results and their validity are discussed. Furthermore, dynamic fluid flow for a viscous fluid in a pipe as a benchmark for the bulk fluid model is presented. To represent dynamic fluid behavior in the FPMI problem, MED description was chosen and to demonstrate its ability to represent dynamic fluid behavior in a problem with moving boundary conditions, which is mostly the case in FPMI problems, a model with periodic motion defined for the boundaries is represented in Chapter 5. Furthermore, a numerical solution for the FPMI problem is demonstrated as an example and the outcome of the simulation regarding precision, errors, limits and applicability of the staggered solution of the FPMI problem is explored. Moreover, with the result of applied CSS algorithm in PANDAS environment for the FPMI problem at hand, a consideration of costs and benefits of the staggered solution implementation for the FPMI problem in PANDAS is discussed.

The materials and information provided by this master thesis can be useful in a handful of different aspects of continuum mechanics, biomechanics, and specifically fluid-porous-media interaction modeling. Throughout this work, a dynamic model for Navier–Stokes governed fluid flow was created in PANDAS environment which can be useful for future deeds regarding the modeling of a bulk viscous/inviscid fluid. This model comes handy in most approaches toward fluid interaction with structures, porous medium, and another fluid medium. The formulation regarding the bulk fluid for this master thesis were developed in a general approach and applied to 3-d cases, although, the model implemented in PANDAS was prepared for 2-d problems due to simplicity and to avoid unnecessary time and energy consumption. Different simplifying assumptions regarding the FPMI phenomenon was taken into account among which the most not so realistic one was neglecting the mass transfer between the bulk fluid and porous medium. Considering this assumption, three major changes are required in the generated formulation presented in this master thesis

1. The translation of the results of one subsystem as the boundary conditions for the other subsystem will differ regarding the fact that fluid pressure at the boundary will act as the pore fluid pressure for the porous media.

2. Mass exchange between two subsystem changes the mixture volume balance formulation for the porous media.

3. Seepage velocity \( w_F \) is no longer equal to zero on the wet boundary for the porous medium subsystem. Therefore, at each time step seepage velocity should be calculated. However, the pore fluid velocity \( v_F \) is equal to bulk fluid velocity \( v_L \) which makes the calculation of the seepage velocity feasible.
In this research, CSS algorithm was chosen as a mean for coupling fluid and porous medium. However, CSS is not the sole algorithm for staggered solution of fluid-structure or fluid-porous-media coupling. Other coupling algorithms such as Improves Serial Staggered (ISS) procedure [9] and Improved Parallel Staggered algorithm (IPS) [9] can also be implemented in PANDAS environment by utilizing the models developed in this master thesis.

Another aspect of this work which has the potential to be further investigated is the consideration of thermo-mechanical coupling in both bulk fluid and porous medium. This consideration can be useful in researches related to ice-formation in FPMI simulations and also in case of fracking simulation, in which there is phase change for fluid due to temperature changes.

Furthermore, one of the main applications of this work can be the numerical solutions of problems with big differences in the scales of their models e.g. modeling of the geothermal powerplants. In these problems instead of the coupling of two completely different environments, the only difference is in the scales which makes it extremely costly in case of using one model for the whole problem. The results of this work can be used to couple different parts of these problems with different scales and transfer the results between these partitions which would decrease the computation costs significantly.
Bibliography


Published Preprints

http://www.nupus.uni-stuttgart.de

2007/1  Cao, Y. / Eikemo, B. / Helmig, R.: Fractional flow formulation for two-phase flow in porous media

2007/2  Korteland, S.-A., The average equilibrium capillary pressure-saturation relationship In two-phase flow in porous media

2008/1  Helmig, R. / Weiss, A. / Wohlmuth, B.: Variational inequalities for modeling flow in heterogeneous porous media with entry pressure

2008/2  Cao, Y. / Helmig, R. / Wohlmuth, B.: Convergence study and comparison of the multipoint flux approximation L-method

2008/3  van Duijn, C.J. / Pop, I.S. / Niessner, J. / Hassanzadeh, S.M.: Philip’s redistribution problem revisited: the role of fluid-fluid interfacial areas

2008/4  Niessner, J. / Hassanzadeh, S.M.: Modeling kinetic interphase mass transfer for two-phase flow in porous media including fluid–fluid interfacial area


2008/6  Cao, Y. / Helmig, R. / Wohlmuth, B.: Geometrical interpretation of the multipoint flux approximation L-method

2008/7  Vervoort, R.W. / van der Zee, S.E.A.T.M.: Simulating the effect of capillary flux on the soil water balance in a stochastic ecohydrological framework

2008/8  Niessner, J. / Hassanzadeh, S.M.: Two-phase flow and transport in porous media including fluid–fluid interfacial area


2008/10 Wolff, M.: Comparison of mathematical and numerical models for two phase flow in porous media


2008/12 Cao, Y. / Helmig, R. / Wohlmuth, B.: Convergence of the multipoint flux approximation L-method for homogeneous media on uniform grids

2008/13 Ochs, S.O.: Development of a multiphase multicomponent model for PEMFC

2008/14 Walter, L.: Towards a model concept for coupling porous gas diffusion layer and gas distributor in PEM fuel cells

2009/1  Hægland, H. / Assteerawatt, A. / Helmig, R. / Dahle, H.K.: Streamline approach for a discrete fracture-matrix system

2009/3 Heimann, F.: An unfitted discontinuous Galerkin method for two-phase flow

2009/4 Hilfer, R. / Doster, F.: Percolation as a basic concept for macroscopic capillarity

2009/5 van Noorden, T.L. / Pop, I.S. / Ebigo, A. / Helmig, R.: An effective model for biofilm growth in a thin strip

2009/6 Baber, K.: Modeling the transfer of therapeutic agents from the vascular space to the tissue compartment (a continuum approach)

2009/7 Faigle, B.: Two-phase flow modeling in porous media with kinetic interphase mass transfer processes in fractures

2009/8 Fritz, J. / Flemisch, B. / Helmig, R.: Multiphysics modeling of advection-dominated two-phase compositional flow in porous media

2009/9 Støverud, K.: Modeling convection-enhanced delivery into brain tissue using information from magnetic resonance imaging


2010/1 Rosenbrand, E.: Modelling biofilm distribution and its effect on two-phase flow in porous media

2010/2 Schöniger, A.: Parameter estimation by ensemble Kalman filters with transformed data


2011/1 Linders, B.: Experimental investigations on horizontal redistribution

2011/2 Rau, M.T.: Geostatistical analysis of three-dimensional hydraulic conductivity fields by means of maximum Gauss copula

2011/3 Kraus, D.: Two phase flow in homogeneous porous media - The role of dynamic capillary pressure in modeling gravity driven fingering

2011/4 Brugman, R.: Dimensionless analysis of convection enhanced drug delivery to brain tissues

2011/5 Sinsbeck, M.: Adaptive grid refinement for two-phase flow in porous media


2012/1 Köppl, T. / Wohlmuth, B. / Helmig, R.: Reduced one-dimensional modelling and numerical simulation for mass transport in fluids

2012/2 Kumar, K. / Pop, I.S. / Radu, F.A.: Convergence analysis for a conformal discretization of a model for precipitation and dissolution in porous media
2012/3 Hommel, J.: Modelling biofilm induced calcite precipitation and its effect on two phase flow in porous media

2012/4 Estrella, D.: Experimental and numerical approximation methods for zero-valent iron transport around injection wells

2012/5 Heimhuber, R.: Efficient history matching for reduced reservoir models with PCE-based bootstrap filters

2012/6 Kissling, F. / Karlsen, K.H.: On the singular limit of a two-phase flow equation with heterogeneities and dynamic capillary pressure

2012/7 Fritz, S.: Experimental investigations of water infiltration into unsaturated soil - Analysis of dynamic capillarity effects

2012/8 Strohmer, V.: Numerische Analyse von nahezu parallelen Strömungen in porösen Medien

2012/9 Kissling, F. / Rohde, C.: The computation of nonclassical shock waves in porous media with a heterogeneous multiscale method: The multidimensional case

2012/10 Fetzer, T.: Numerical analysis of the influence of turbulence on exchange processes between porous-medium and free flow

2012/11 Schröder, P.: A response surface bootstrap filter to calibrate CO₂ injection models


2013/2 Köppel, M.: Flow modelling of coupled fracture-matrix porous media systems with a two mesh concept

2013/3 van Helvoort, M.: Upscaling of processes involving rough boundaries

2013/4 Redeker, M. / Haasdonk, B.: A POD-EIM reduced two-scale model for crystal growth

2013/5 Vogler, D.: A comparison of different model reduction techniques for model calibration and risk assessment

2014/1 Song, N.: Investigation of a decoupling scheme for the modeling of reactive transport

2014/2 Aydogdu, A.B.: Phase field modelling of critical shear band evolution in granular media on the basis of a microporous medium theory

2014/3 Becker, B.: Investigation of error estimates for cell centered finite volume schemes: Analysis and improvement of grid adaptation strategies in DuMux

2014/4 Moghaddam, N.D.: Sorption of methane and ethane on Belgian black shale using a manometric setup

2014/5 Schwenck, N. / Flemisch, B. / Helmig, R. / Wohlmuth, B.: Dimensionally reduced flow models in fractured porous media: crossings and boundaries

2014/6 Redeker, M. / Pop, S. / Rohde, C.: Upscaling of a tri-phase phase-field model for precipitation in porous media

<table>
<thead>
<tr>
<th>Year</th>
<th>Authors</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>2014/8</td>
<td>Radu, F.A. / Nordbotten, J.M. / Pop, I.S. / Kumar, K.</td>
<td>A robust linearization scheme for finite volume based discretizations for simulation of two-phase flow in porous media</td>
</tr>
<tr>
<td>2014/9</td>
<td>Koch, T.</td>
<td>Coupling a vascular graph model and the surrounding tissue to simulate flow processes in vascular networks</td>
</tr>
<tr>
<td>2014/10</td>
<td>Fritsch, M.</td>
<td>Analysis of Processes and Properties at a Porous-Medium Free-Flow Interface with the Application of Evaporation</td>
</tr>
<tr>
<td>2014/11</td>
<td>Seitz, G.</td>
<td>A conceptional approach to the effect of shear failure on the stress field in porous media</td>
</tr>
<tr>
<td>2014/12</td>
<td>Gläser, D.</td>
<td>Spatial model coupling for the simulation of CO2 injection scenarios in deep saline aquifers</td>
</tr>
<tr>
<td>2015/2</td>
<td>Bause, M. / Radu, F. A./ Köcher, U.</td>
<td>Error analysis for discretizations of parabolic problems using continuous finite elements in time and mixed finite elements in space</td>
</tr>
<tr>
<td>2015/3</td>
<td>van Duijn, C. J. / Cao, X. / Pop, I. S.</td>
<td>Two-phase flow in porous media: dynamic capillarity and heterogeneous media</td>
</tr>
<tr>
<td>2015/4</td>
<td>Pop, I. U. / Bogers, J. / Kumar, K.</td>
<td>Analysis and upscaling of a reactive transport model in fractured porous media involving a nonlinear transmission condition</td>
</tr>
<tr>
<td>2015/5</td>
<td>Jambhekar, V. A. / Mejri, E. / Schröder, N. / Helming, R. / Shokri, N.</td>
<td>Kinetic approach to model reactive transport and mixed salt precipitation in a coupled free-flow-porous-media system</td>
</tr>
<tr>
<td>2015/6</td>
<td>List, F. / Radu, F. A.</td>
<td>A study on iterative methods for solving Richards’ equation</td>
</tr>
<tr>
<td>2015/8</td>
<td>Hommel, J. / Lauchnor, E. / Gerlach, R. / Cunningham, A. B. / Ebigbo, A. / Helming, R. / Class, H.</td>
<td>Investigating the influence of the initial biomass distribution and injection strategies on biofilm-mediated calcite precipitation in porous media</td>
</tr>
<tr>
<td>2015/9</td>
<td>Weishaupt, K.</td>
<td>Numerical modeling of steam chamber build-up guided by hot-water pre-injection</td>
</tr>
<tr>
<td>2015/10</td>
<td>Asgharzadeh, P.</td>
<td>Fluid-Porous-Media Interaction Modeling with Staggered Solution Schemes</td>
</tr>
</tbody>
</table>