INTERNATIONAL JOURNAL OF HUMAN COMPUTING STUDIES

Vol. 6 | Issue 2 **| pp. 24**-32 **| e-ISSN:** 2615 – 8159 **| p-ISSN:** 2615 - 1898
Available online @ https://journals.researchparks.org/index.php/IJHCS

research parks publishing

The Subject of This Study is the Multiphase Flow of a Compressible Liquid in a Porous Medium, Specifically Focusing on Classification

Bustanov A. Khudaykul*

[*]Samarkand State University in Samarkand, Uzbekistan.

Abstract: Modeling the flow of two-phase compressible fluids through porous media is very pertinent to a broad spectrum of physical and technical applications. The study focuses on reservoir modeling and oil and gas production, which require the use of advanced numerical methods to ensure efficiency. The objective is to achieve a numerical solution to this model by integrating finite element and finite volume approaches. This involves generating velocity values at the boundary of the finite volume grid cells based on point pressure values at KE nodes.

Keywords: Modeling, filtration, compressibility, porosity, medium, oil, fluid, buoyancy, saturation, simulation, eddies, pressures.

Citation: Bustanov A. Kh. The Subject of This Study is the Multiphase Flow of a Compressible Liquid in a Porous Medium, Specifically Focusing on Classification. *International Journal for Human Computing Studies,* **2024**, *6(2)*, 24-32.
https://doi.org/10.31149/ijhcs.v6i2.5 249

Received: 12 March 2024
Revised: 29 March 2024
Accepted: 28 April 2024
Published: 13 May 2024

1. Introduction

The study of multiphase flow in porous media is a highly dynamic area of research. This discipline investigates various practical fluid dynamics scenarios, including groundwater contamination, oil and gas extraction, and CO2 storage in geological formations. Due to the significant interest in multiphase flow in porous media, numerous studies have been conducted in this field. Some notable references, including [1, 10, 11], have been published on this topic. This work focuses on the mathematical and numerical modeling of underground flow in water-oil systems. In this scenario, the fluid in motion often consists of a blend of oil, water, and gas flowing through a permeable substance as a result of the interconnected network of pores within the geological medium. The prevailing mathematical models used to depict these types of problems consist of a set of parabolic partial differential equations. These equations describe the pressure and saturation of the various phases present in the process, specifically known as the Black-Oil model.

Several modeling tools employ finite difference methods or finite element approaches to solve mathematical models [10]. The user mentions the use of finite element methods, sometimes known as [10]. Additional formulations of finite volume methods and combinations of finite element and finite volume approaches can be found in existing literature. The model explores strategies, as seen in Eq. [4], that includes transforming a parabolic FVM system into a system consisting of an elliptic equation for pressure and a hyperbolic equation for saturation. The second equation bears resemblance to the widely recognized Buckley-Leverett equation, which was initially presented in reference [9]..

The theoretical analysis of this formulation was developed in reference [2]. The intriguing aspect of this study involves the handling of the two-phase model by converting it into an elliptic-hyperbolic system. The paper encompasses both the analytical investigation and the numerical solution [2]. In addition, a numerical solution of the simplified model is executed in reference [2]. The authors in [12] demonstrate the local existence and uniqueness of the classical solution for the hyperbolic-elliptic system that arises in the modeling of oil production processes. They achieve this by utilizing the Artzel-Askoli theorem. Another significant contribution on the presence and uniqueness of the solution in models of filtration of immiscible fluids in porous media.

The elliptic equation, which represents pressure, and the hyperbolic equation, which represents saturation, are interconnected through the inclusion of saturation in the mobility coefficients of both equations. The numerical strategy employed in this model involves solving the pressure equation, which is the elliptic component of the model, using the finite element method. Additionally, the saturation equation, which is the hyperbolic part, is solved using a finite volume scheme. The rationale behind employing these two distinct approaches in the same problem stems from the fact that finite element methods were specifically designed for addressing elliptic (and parabolic) problems. However, they prove to be highly inadequate, particularly in their conventional formulation, when confronted with hyperbolic problems, especially those involving discontinuous solutions. Finite-volume techniques are effective in these specific scenarios as they can accurately propagate discontinuous solutions. This model calculates the pressure at the nodes where the domain is divided into smaller sections, and these nodes are then used as borders between the sections in the finite volume scheme. It is important to understand that the finite volume scheme relies on the use of integral mean values of the solution, namely the saturation in this case. Given the dependence of velocities at the cell borders on the pressure gradient, it is necessary to apply a reconstruction method by Darcy's law. This model employs a piecewise linear reconstruction, taking into account the scenario that minimizes the absolute value of the slope. An explicit Euler technique is used to perform the temporal integration for the hyperbolic equation. It is important to understand that this process is distinct from the standard IMPES method. In the classical method, both equations evolve and a pressure implicit scheme is employed, whereas an explicit formulation is used to address the saturation problem [4]. The study presents results for the single-phase scenario, where a parabolic equation is used to model pressure. Additionally, the study also presents results for pressure and saturation in the two-phase scenario. An elastic porous material saturated with a compressible fluid can be described using a collection of physical variables based on a specific parameter. The fluid flow in a porous medium can be conceptualized as a combination of two phases, namely liquid and elastic porous. Each phase can be distinguished by its unique set of state parameters. It is important to mention that in this context, we are examining processes where temperature variations are insignificantly tiny. Additionally, we are utilizing the isoentropic approximation of the model. The mixture element is characterized by the following state parameters:

α1 - the volume fraction of the elastic medium; ϱ - the mass density of the mixture;

c1 - the mass fraction of the elastic medium;

in - the velocity of the mixture;

wi - the relative velocity of the elastic porous medium concerning the liquid; Fij - the elastic deformation gradient of the mixture.

The liquid's volume and mass fractions can be determined by calculating $\alpha_2 = 1 - \alpha_1$, and $c_2 = 1 - c_1$, respectively. The parameters that describe the state of the mixture are connected to the state parameters of the phases through specific relationships.

$$\rho = \alpha_1\rho_1 + \alpha_2\rho_2, \quad c_1 = \alpha_1\rho_1/\rho, \quad c_2 = \alpha_2\rho_2/\rho, \quad u^i = c_1u_1^i + c_2u_2^i, \quad w^i = u_1^i - u_2^i,$$

The variables ϱ1 and ϱ2 represent the mass densities of the elastic porous media and the liquid, respectively. These variables also correspond to the velocities of the respective substances.

Here we propose a model of two-dimensional test problems to validate the model. Given that the constitutive equations constitute a hyperbolic system of conservation laws, it is possible to directly utilize the conventional WENO-Runge-Kutta numerical method [9] to solve the two-dimensional form of these equations. The 2D version of the system of equations comprises 11 equations, which can be formally expressed as:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial x_1} + \frac{\partial \mathbf{G}(\mathbf{U})}{\partial x_2} = \mathbf{Q}(\mathbf{U}),$$

Let U represent the vector of conservative variables. F(U) and G(U) denote the fluxes, whereas Q(U) represents the vector of source terms. Here, we utilize the identical approach for solving the initial boundary value problem. The Lax-Vendroff flux is employed for approximating the flux within the intercells, while the characteristic decomposition and linearized boundary conditions are utilized for estimating the flux at the border. It is important to consider that the pressure relaxation occurs immediately in this case, given that the size of the pore space is relatively modest. The numerical approach for handling the immediate pressure relaxation involves adjusting the phase volume fraction $\alpha 1$ and densities $\varrho 1$, $\varrho 2$ by solving the algebraic equation p1 = p2 after each time step for every grid cell.

The equation of state of an ideal gas:

Where: $p \cdot V_M = R \cdot T,$

p is the pressure,

$V_м$ - molar volume,

R is the universal gas constant

T - absolute temperature, K.

Physical model

An oil reservoir is a geological formation that contains a mixture of oil, water, and gas. Initially, the hydrocarbon reservoir is in equilibrium and contains reservoir fluids such as gas, oil, and water separated by gravity under no-flow conditions.

When a well is drilled reaching the upper reservoir formation, this equilibrium state is immediately disrupted, hence the pressure at that particular location drops, gradually propagating in all radial directions from the well zone to the entire reservoir formation. The study of fluid motion in porous media is very complicated. Therefore, it is necessary to introduce some simplifying assumptions.

Boundary conditions

The following hypothesis is established in this model, given below.

- The flow occurs in one dimension.

- The fluids do not mix and their composition remains constant in time, so there is no mass transfer between phases.

- The system is isothermal.

- Capillary pressure is neglected.

- Horizontal flow is assumed, so the effect of gravity is neglected.

Blackoil model. The thermodynamic black oil model is one of the most widely used models in oil reservoir modeling.

A hydrocarbon is often characterized by two components: a heavy component, known as oil, and a light component, known as gas. These components maintain a stable composition over time. Additionally, it is thought that there is no mass transfer between the phases, meaning that they do not mix. Under reservoir conditions, both components have the potential to dissolve completely or partially, depending on the pressure and temperature in the reservoir. This results in the formation of one or two phases, specifically liquid and gas. The Blackoil equations have the following components. - Conditions of thermodynamic equilibrium.

An equation of state characterizes the fluid by its fundamental physical properties, as expressed by defining equations.

The principle of Darcy's law governs the volume flow rates of a system, specifically the

conservation of momentum. The mass conservation equation is applied to each component individually. The Black Oil model characterizes the reservoir under constant temperature conditions, where the fluid properties are solely determined by the pressure behavior within the reservoir. Hence, a table that illustrates the relationship between the change in data and the related pressure change over time would suffice. This table will specify the PVT condition for each phase [7].

The saturation pressure, represented as point 2 in the diagram, corresponds to the specific condition at which the initial gas bubble is generated during the process of saturation. Below the saturation point, the reservoir has two hydrocarbon phases: a liquid phase known as gas-saturated oil, and a gas phase which is the released dissolved gas. Once the saturation point is surpassed, the oil starts to emit dissolved gas, which then maintains a consistent concentration throughout the process. This gas is classified as degassed oil since the presence of dissolved gas does not have any impact on its behavior at a consistent temperature, which is always maintained below the critical point. When the pressure in the reservoir exceeds the saturation threshold, there is only one phase of hydrocarbon, specifically liquid oil. This oil is referred to as degassed oil or undersaturated oil because it is constantly exposed to pressure levels that exceed the saturation point, preventing the release of any gas from the oil [9].

Constructing a closed-loop model. The research focuses on a one-dimensional two-phase model, which is applicable in the field of reservoir modeling and oil recovery. Within the model, the reservoir contains solely oil and water as the active phases. The objective of this model is to elucidate the process of immiscible displacement of oil, wherein oil is propelled towards a producing well due to the injection of water into the reservoir. Oil and water are immiscible. The purpose of drilling an injection well is to uphold elevated pressure in the reservoir by introducing fluids, such as water or gas. This enables the displacement of hydrocarbons towards the well when the existing pressure difference between the reservoir and the wellbore is insufficient for uninterrupted hydrocarbon production. Developing a mathematical model that accurately depicts the fluid flow behavior in an oil reservoir is crucial. This model consists of a collection of nonlinear partial differential equations (PPCMs) [10, 11]. Optimizing the computations for fluid simulation is crucial to ensure both speed and accuracy. To accomplish this, one must select a maximum time interval that is sufficiently small to avoid compromising the accuracy of the simulation. The Courant-Friedrichs-Lewy criterion (CFL) is typically employed to determine the appropriate time interval Δt. According to this criterion, Δt should be selected in such a way that the displacement of the fluid particle does not exceed one spatial grid step Δh within Δt.

$\Delta t = \frac{\Delta h}{\vec{u}_{max}}$,

where $\vec{u}_{max}$ – is the maximum velocity in the velocity field.

One of the formulas for calculating $\vec{u}_{max}$ – looks like [3]:

$\vec{v}_{max} = \max(|\vec{u}|) + \sqrt{\Delta h|\vec{F}|}$,

where $\max(|\vec{u}|)$is the largest modulo value of the velocity in the grid, and $\vec{F}$ is the forces applied to the fluid.

However, often 100% accuracy of the simulation is not that important, so it is considered that it is possible toobtainedhe time interval by a factor of k_{CFL}obtained by applying the criterion:

$\Delta t = k_{CFL}\frac{\Delta h}{\vec{u}_{max}}$.

For example, in one of the papers [8], researchers were able to increase the value of the CFL criterion by a factor of 5 without harming the realism of the simulation.

To explain what advection is, we can use the informal question "How will some quantity Q, whose changes are measured in our grid, change in Δt?". Formally, advection on the (n+1)-th time interval is described by the following function:

$Q^{n+1} = advect\,(Q^n, \Delta t, \frac{\partial Q^n}{\partial t})$.

Hereinafter we will assume that Q^n is the value of Q at the n-м time interval. The algorithm for calculating advection is given below.

For each grid cell with indices i, j, k

Calculate $-\frac{dQ}{dt}$

Calculate the position of $Q_{ijk}$and store it in $\vec{X}$

$$\vec{X}_{prev} = \vec{X} - \frac{dQ}{dt}\Delta t$$

Find the grid node nearest to $\vec{X}_{prev}$, assign it the value Q_{ijk}

$Q = Q^{n+1}$

Then the closed model has the form:

$$\left|\left|P_{ij}\right|\right| = \begin{vmatrix} q_{00} & q_1 & \dots & q_i & \dots & q_n & 0 \\ R_1 & q_{11} & \dots & 0 & \dots & 0 & \overline{R_1} \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ R_n & 0 & \dots & 0 & \dots & q_{nn} & \overline{R_n} \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{vmatrix}$$, where, $q_{00}=1-q_{\Sigma}$

This algorithm is a realization of Euler's explicit method, which is a numerical technique for solving differential equations. The orange point is located within the cell where the calculation of advection is to be performed. This is accomplished based on the velocity field. Incorporating pressure in simulations is essential for several reasons. Firstly, it is necessary to maintain a constant volume of the fluid, taking into account the fact that the fluid is incompressible and ensuring that the rule of conservation of mass is not violated. To achieve this objective, the continuity equation has the following form:

$\mathrm{div} = 0.$

The meaning of the equation is that in all faces (all_faces) of each cell the flux is the same and equal to zero:

$$\sum_{all\_faces} flux = 0$$

To solve the problem, we need to solve the following equation:

$$u^{n+1} = u^n - \frac{1}{\rho}\nabla \mathrm{p} \qquad (1)$$

where the unknowns are u^{n+1} и ∇p.

If the fluid is incompressible, the divergence is zero. This can be written as:

$$\nabla \cdot u^{n+1} = 0 \qquad (2)$$

Given condition (2), which we apply to equation (1), we obtain the Poisson equation:

$\frac{1}{\rho}\nabla^2 p = \nabla u^n$.

To solve this equation, the finite difference method is used - a system of algebraic equations of the form is constructed based on the available differential problem:

$\mathrm{A}p = d,$

where A is some matrix of coefficients, p is to be found. The system is most often solved by the conjugate gradient method, which is an efficient iterative method. After solving the Poisson equation, p can be substituted into equation (1) and the updated velocity field can be calculated.

Special attention should be paid to the grid boundary nodes, which can be of two types: free surface (*free surface - the fluid can move freely, p=0),* solid wall (*solid wall* - the fluid cannot "flow" through it, and the velocity at the boundary is equal to the velocity at the rigid boundary in the direction of the normal n. For this purpose, the Neumann problem is solved: $u_{boundary} \cdot n = u_{solid} \cdot n$).

If the pressure at the free surface is correctly described, the fluid will "splash" in the air. Rigid boundaries are the walls of the describing cube.

Choosing values for the equations of state experiments The expression $\langle\langle \phi, \psi\rangle\rangle_{\mathbb{C}} = \bar{\phi}_1\psi_1 + \bar{\phi}_2\psi_2$ represents the Hermite form on $\mathbb{C}^2$in this model. The scalar product on the complex vector space $\mathbb{C}^2$is represented as $\langle\langle \phi, \psi\rangle\rangle_{\mathbb{R}} = Re(\langle\langle \phi, \psi\rangle\rangle_{\mathbb{C}})$. On the real vector space $\mathbb{R}^3$, the scalar product is denoted as $\langle u, v\rangle$;. The notation $|\psi|^2 = \langle\langle \phi, \psi\rangle\rangle_{\mathbb{R}}$_R is used to indicate the quadratic norm in the streamwise direction. The symbol ξ represents the divergence. The decreased Planck constant,

commonly known as the Dirac constant, is denoted by the symbol ħ. This model employs computations based on the theory of Discrete Exterior Calculus (DEC)[1]. The theory emerged as a result of the need to develop a functional framework for working with theories of different types of physical fields and various differential calculations. Generally, the DEC system bears resemblance to vector calculus, but with the inclusion of novel terminology. As an illustration, in this context, a scalar field is referred to as a 0-form. Allow us to review the fundamental principles of differential geometry as applied to surfaces (refer to Figure 5). A vector v is defined by its magnitude and orientation. The value α(v) represents the magnitude of the component of the vector v in the direction α. A (differential) 1-form is a linear function that maps a vector to a scalar quantity.

Measurement of the relative permeability of phase

The following algorithms utilize the Fast Fourier Transform (FFT) and its inverse, the Inverse Fast Fourier Transform. The authors of the research utilize the Fourier domain to conduct a portion of their calculations, employing the Fast Fourier Transform (FFT) and its inverse. Figure 6 illustrates the determination of relative phase permeability. The fluid is permeable on the far right, exhibits viscosity in the center, and is entirely viscous on the far left. The primary rationale for utilizing frequency domains is that numerous operations may be executed with greater precision in these domains [3]. In image processing, the Fast Fourier Transform (FFT) is employed to calculate convolution, which is then transformed into multiplication. Fluid simulation involves representing the velocity field as the combination of the mass-conserving field and the gradient field. Calculations are conducted on a three-dimensional grid, as shown in Figure 1:

$$\mathcal{V} = \{0, \dots, \mathcal{N}_x - 1\} \times \{0, \dots, \mathcal{N}_y - 1\} \times \{0, \dots, \mathcal{N}_z - 1\}$$

$\mathcal{N}_x$, $\mathcal{N}_y$, $\mathcal{N}_z$ – are the grid sizes along the corresponding axes. The vertices v∈V store the values of the wave function $\psi_v \in \mathbb{C}^2$, the pressure $q_v \in \mathbb{R}$, and the divergence $\xi_v \in \mathbb{R}$. The algorithm uses the discrete velocity as the weight of the directed grid edges $vw \in \mathcal{E}$:

$\eta_{vw} = \hbar \, arg\langle \psi_v, \psi_w \rangle_{\mathbb{C}}$,

which is stored in staggered order in the vertices of the grid.

Discrete divergence is expressed through the formula:

$$\xi_v = \frac{1}{V_v} \sum_{vw \in \mathcal{E}} \frac{A_{vw}}{l_{vw}} \eta_{vw},$$

where A_{vw} is the doubled area of the face, $l_{vw}$is the length of the mesh edge, and V_v is the doubled volume of the cell.

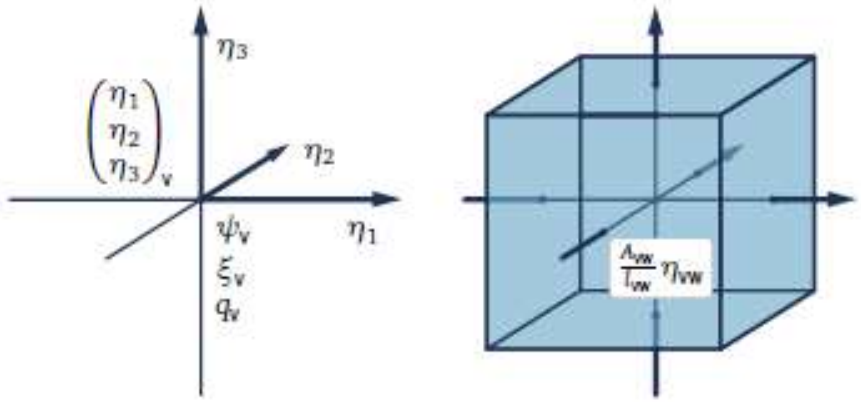


Fig. 1 - Three-dimensional grid for calculations

The input is the initial value of the functio $\psi^{(0)}$, a positive time step dt, and the quantization power ħ.

Here the splitting operator is used, normalization is performed, and the pressure is calculated. Obstacles and buoyancy (taking into account gravitational forces) are also taken into account here. The initial parameters were found by the authors of the method by fitting.

Mathematical description of initial and boundary conditions

The basic algorithm for compressible fluid flow in a porous medium - initial condition - is presented below.

Input: $\psi^{(0)}$, dt, ħ //initial condition and parameters
For j← 0,1,2,...
$\psi^{\text{время}time} \leftarrow Schrödinger(\psi^{(j)}, dt, ħ)$
$\psi^{time} \leftarrow \frac{\psi^{time}}{|\psi^{time}|}$ //normalization
$\psi^{(j+1)} \leftarrow PressureProject(\psi^{time})$

This function converts the matrix (grid) to a diagonal form. For this purpose, the FFT is used (for values on the grid faces - DCT).

In this case λ_v are the eigenvalues of the three-dimensional continuous Laplace operator , which is a boundary condition.

The function Schrödinger(ψ, dt, ħ)
$\hat{\psi} \leftarrow FFT3D(\psi)$
$\hat{\psi} \leftarrow e^{i\lambda dt\frac{ħ}{2}}\hat{\psi}$
Back $InvFFT3D(\hat{\psi})$

Mathematical description of the equation for pressure

Let us make a mathematical description of the equation for pressure.

Pressure can affect a compressible fluid in three states:

[153] State "0": no pressure is exerted on the fluid;

[154] State "1": pressure begins to compress the fluid.

[155] State "2": pressure has compressed the fluid to its physical limit.

[156] State "3": the pressure stops acting on the fluid.

Then given the states: $P_0(0) = 1; P_1(0) = P_2(0) = 0$ The mathematical description of the pressure equation will look as follows:

$$P_0(1) = 1 - q_\Sigma; P_1(1) = q_1; P_2(1) = q_2; P_3(1) = 0,$$
$$P_0(2) = (1 - q_\Sigma)^2 + q_1R_1 + q_2R_2;\ P_1(2) = (1 - q_\Sigma)q_1;$$
$$P_2(2) = (1 - q_\Sigma)q_2; P_3(1) = q_1\overline{R_{13}} + q_2\overline{R_{23}}$$

This function calculates the pressure of the compressible fluid within the confining volume, including the pressure exerted on the walls. This portion of the algorithm is executed using wave functions, as opposed to the conventional approach which often employs the direct Euler technique.

Function $PressureProject(\psi)$
For all $vw \in \mathcal{E}$
$\tilde{\eta}_{vw} = arg\langle\psi_v, \psi_w\rangle_{\mathbb{C}}$
For all $v \in \mathcal{V}$
$\xi_v = \frac{1}{V_v}\sum_{vw\in\mathcal{E}}\frac{A_{vw}}{l_{vw}}\tilde{\eta}_{vw}$
$\hat{\xi} \leftarrow FFT3D(\xi)$
$\hat{\xi} \leftarrow \hat{\xi}\begin{cases}\tilde{\lambda}^{-1}, & if\tilde{\lambda} \neq 0\\ 0\end{cases}$
$q \leftarrow InvFFT3D(\hat{\xi})$

Back $e^{-iq}\ \psi$ // calibration transformation

The eigenvalues of the discrete Laplace operator are utilized in this scenario. The gauge transform operation should be described in isolation. The wave function is a complex entity, yet the measurable quantities resulting from operations on wave functions are real (as expected, since all quantities in the actual world can be measured using real numbers). Quantum mechanics remains unchanged when the wave function is multiplied by a complex number modulo ne ($e^{i\alpha}$), where α represents the phase of the wave function. This means that the theory is invariant concerning global phase rotations, and the symmetry is preserved. **Numerical scheme** refers to a method or algorithm used to solve mathematical problems using numerical calculations. The equation employs local gauge transformations, which lack invariance - the function might undergo varying phase changes at different spatial positions. To restore invariance, it is necessary to introduce a new physical field. Therefore, the wave function's symmetry remains intact [2].

The 1-form velocity function is computed using three input parameters: the values of two wave functions, psi1, and psi2, as well as the Dirac constant *bar*. The *conj* function is utilized to acquire the complex conjugate of an integer. Return the calibrated transformation of e^(-qi) ψ. Here, the eigenvalues of the discrete Laplace operator are utilized.

It is worthwhile to provide a separate explanation of the functioning of gauge transform. The wave function is a complex number, but the measured quantities resulting from operations on wave functions are real (as expected, as all quantities in the actual world can be measured using real numbers). Quantum mechanics remains unchanged when the wave function is multiplied by a complex number modulo one (e^iα), where α represents the phase of the wave function. This means that the theory is invariant under global phase rotations, and the symmetry is preserved. Numerical scheme refers to a method or algorithm used to solve mathematical problems using numerical calculations. The equation employs local gauge transformations, which lack invariance - the function might vary by different phases at distinct positions in space. To restore invariance, it is necessary to introduce a new physical field. Therefore, the wave function's symmetry remains unchanged [2].

The 1-form velocity function is computed using three input parameters: the values of two wave functions, psi1, and psi2, as well as the Dirac constant bar. The conj function is utilized to acquire the complex conjugate of an integer.

Figure 8 illustrates the image of the initial number z and its complex conjugate z^{-}. These numbers are positioned symmetrically about the real axis. The % operator is used to do element-by-element multiplication of matrices. The mod(x,y) function calculates the modulus of each element in the vector x concerning the number y. The variables *iX*, *iY*, and *iZ* represent one-dimensional arrays. The variables *resX*, *resY*, and *resZ* indicate the number of cells in each change. The variables *sliceX*, *sliceY*, and *sliceZ* are used to retrieve the grid slice in the respective dimensions. The incompressible flow simulation algorithm operates using two input parameters, namely the values of two wave functions, psi1 and psi2. The variable "*schrodingerMask"* represents the Fourier coefficient used to solve the equation. In addition, this approach utilizes the three-dimensional Fast Fourier Transform (*FFT*) and the inverse *FFT (IFFT).* The function *fftshift3D(X)* reorganizes the output arrays of the FFT by positioning the zero frequency at the center of the spectrum. To provide further clarification, below are a few illustrative instances of how the *fftshift* function operates. For instance, in the case of a one-dimensional array, the outcome of the array transformation will involve reorganizing its left and right portions cyclically. When applied to a two-dimensional array, *fftshift* exchanges the positions of the first quadrant with the third quadrant, and the second quadrant with the fourth quadrant (see to Figure 2).

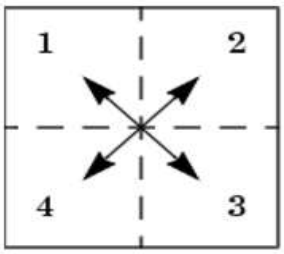
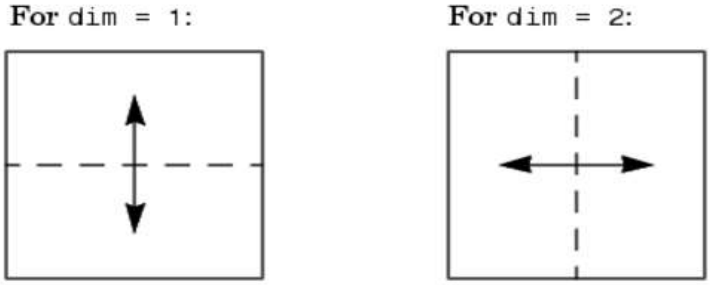


Fig. 2 - Schematic of how *fftshift* works for a matrix

For arrays of higher dimensions, *fftshift* performs a move of two half-spaces in each dimension



(Fig. 3).

Fig. 3 - Principle of moving half-spaces in *fftshift* on the example of a matrix

Then the numerical scheme appears in the form as:

$$\left|\left|P_{ij}\right|\right| = \begin{vmatrix} 1-q_\Sigma & q_1 & q_2 & 0 \\ R_1 & 0 & 0 & \overline{R_{13}} \\ R_2 & 0 & 0 & \overline{R_{23}} \\ 0 & 0 & 0 & 1 \end{vmatrix}, \text{где } q_\Sigma = q_1 + q_2$$

Pressure calculation is based on two input parameters - values of two wave functions *psi1, psi2*. Realizations of the functions *velocityOneForm* and *gaugeTransform* are given in the following paragraphs, div - Hodge operator. The Hodge operator (Hodge star) is a change of basis to an orthogonal basis in the multidimensional case.

Analysis of data and results of computational experiments

The virtual environment solution consists of two projects. The Ecosystem project contains the libraries connected to the project. The *DEC* class contains methods for computing in the *DEC* system, the *ISF* class defines basic functions for simulating a two-phase compressible fluid in a porous medium. *CxCubeVec* and *CubeVec* are overrides for a vector of three-dimensional arrays (with *CxCubeVec* containing complex numbers). The Trackball class implements part of the functionality of the Pez library.

In VisualStudio visualiser, there is a possibility to change some properties of the fluid and features of the implementation, which can be seen from the results of calculations. In particular, these are buoyancy (Liquidbuoyancy), weight (Liquidweight), time interval for rendering (Timestep), grid cell size (Cellsize), obstacle presence (Obstacleexists).

For the Liquidweight parameter it is worth making an important clarification: algorithms and physical formulas usually use density rather than specific gravity of a substance, but one concept is often replaced by another in everyday life and the term "specific gravity" is more understandable for most users. Density and specific gravity are scientifically different terms, but under conditions of approximately constant free fall acceleration g and with negligibly small accelerations that can be neglected in weight calculations (i.e., when simulating conditions on planet Earth), it is possible to use one term instead of the other.

Parameter changes are made using the buttons on the keyboard; if a parameter is changed, the changes are not dynamically applied, i.e., the simulation starts over.

To add or remove an obstacle, use the keyboard shortcut O+Enter. One obstacle is implemented in the program - a sphere under the source of compressible fluid.

The simulated fluid is in a limited volume - a transparent cube of fixed volume. Upon

collision with the transparent walls, the two-phase fluid can be observed to rise up the walls.

Thus, the result of the simulation is parameters for the behaviour of a two-phase compressible fluid in a porous medium, which can be used in the construction of machines and mechanisms to operate under the simulated conditions.

The fundamental assumptions made in developing the model are that momentum transfer between fluids within the porous structure is negligible and a capillary equilibrium condition exists throughout the medium

Conclusion

The established model of biphasic flow in porous media accurately characterizes the hydraulic phenomena and phase interactions that take place. Furthermore, the model exhibits exceptional flexibility by employing distinct conservation equations for each phase inside both the fracture network and the matrix network.

Furthermore, the development of the constitutive and closure relations for the two-fluid model is at a satisfactory level. This text provides a comprehensive analysis of the primary challenges associated with modeling different interfacial transfer circumstances. The transfer of mass and momentum across an interface is directly proportional to the area of the interface and the force that drives the transfer.

References:

[1] Ипатов А.И., Кремнецкий М.И. Геофизический и гидродинамический контроль разработки месторождений углеводородов. - М.: НИЦ «Регулярная и хаотическая динамика»; Институт компьютерных исследований, 2005. - 708 с.

[2] S. Mishra G. Coclite, K. Karlsen, N. Risebro; *A hyperbolic-elliptic model of two-phase flow in porous media-existence of entropy solutions.*, Int. J. Numer. Anal. Mod. 9 (2012), no. 3, 562–583.

[3] W. Kinzelbach; *Groundwater modelling: An introduction with sample programs in basic*, Developments in Water Science, Elsevier Science, 1986.

[4] Кременецкий М.И., Ипатов А.И. Гидродинамические и промыслово-технологические исследования скважин. - М.: МАКС Пресс, 2008. - 476 с

[5] S. Mantica L. Bergamaschi and G. Manzini; *A mixed finite element–finite volume formulationof the black-oil model*, SIAM Journal on Scientific Computing 20 (1998), no. 3, 970–997.

[6] R. J. LeVeque; *Finite volume methods for hyperbolic problems*, Cambridge Texts in Applied Mathematics, Cambridge University Press, 2002.

[7] F. Monkeberg; *Finite volume methods for fluidflow in porous media*, (2012).

[8] D. W. Peaceman; *Fundamentals of numerical reservoir simulation*, Elsevier, 1977.

[9] Зяблицкая Ю. А. Анализ и интерпретация гидродинамических исследований для двухфазного потока (вода-нефть) // Известия ТПУ. 2010. №1. URL: https://cyberleninka.ru/article/n/analiz-i-interpretatsiya-gidrodinamicheskih-issledovaniy-dlya-dvuhfaznogo-potoka-voda-neft (дата обращения: 07.07.2022).

[10] Bustanov Kh. A. Mathematical modeling of two-phase compressible fluid filtration in a porous medium, Journal of Mechanical and Production Engineering (JMPE) ISSN (P): 2278-3512 ISSN (E): 2278-3520 Vol. 12, Issue 2, Dec 2022, 1-10

[11] Bustanov Kh. A. Basic methods, models and calculations for modeling the filtration of a two-phase compressible liquid in a porous medium, Journal of Mechanical and Production Engineering (JMPE) ISSN (P):2278-3512; ISSN (E):2278-3520 Vol. 13, Issue 1, Jun 2023, 13-24

[12] A. Schroll, A. Tveito; *Local existence and stability for a hyperbolic-elliptic system modelingtwo-phase reservoir flow*, Electron. J. Differential Equations 2000 (2000), no. 4, 1–28 (eng).