Generalized cell-centered finite volume methods for flow in porous media with faults

Jérôme Jaffré [**] — **Vincent Martin** [*] — **Jean E. Roberts** [**]

**Andra,*
Parc de la Croix Blanche, 1-7 rue Jean Monnet,
92298 Chatenay-Malabry Cedex, France,
and Inria-Rocquencourt, BP 105,
78153 Le Chesnay cedex, France.

Vincent.Martin@inria.fr

***Inria-Rocquencourt, BP 105,*
78153 Le Chesnay cedex, France.

Jerome.Jaffre@inria.fr — Jean.Roberts@inria.fr

ABSTRACT. *Standard cell-centered finite volume methods are generalized by allowing for higher order polynomials and unstructed meshes while the main physical features are preserved, writing cell equations and cell interactions. This is obtained by using a nonstandard presentation of mixed-hybrid finite elements. The flexibility of the method is demontrated for flow in a porous medium with faults.*

KEYWORDS: *Finite volumes, mixed methods, faults, flow in porous media.*

1. Introduction

Finite volume methods and finite elements methods are generally opposed as competing approximation techniques. However there are ways to look at them which make these methods closer than what is usually thought, and this leads to the construction of generalized finite volume methods. Here we will restrict ourselves to cell-centered finite volumes.

On one hand, cell-centered finite volume methods are widely used by engineers and scientists who have to perform numerical simulations. When constructing such methods, one discretizes the domain, usually with rectangles though not necessarily, and equations are written inside each cell of the discretization. Then relations between cells are written. The solution of the problem is approximated by piecewise constants. Usually flux unknowns are also introduced to describe the intercell relations which insure conservation. This procedure is very close to the physics, and even complicated physical laws can be implemented easily.

However finite element methods also have advantages: they are based on a rigorous mathematical analysis, higher order methods can be obtained by increasing the degrees of the polynomials, and structured as well as unstructured meshes can be used depending on the application that is under study.

In this paper we show how to combine the advantages of these two approximation techniques into one numerical procedure. We retain the point of view of finite volume methods, but we call them generalized since, to approximate the unknowns, we use general discontinuous piecewise polynomial approximations instead of just piecewise constants.

As an illustration we consider as an example a model for incompressible single-phase flow in a porous medium with a large fault, modeled as a hypersurface. The numerical procedure described here follows closely the physical characteristics of the model. It is an extension of that presented in [ALB 99] in that it allows for the case of a geological barrier, that is when the permeability in the fault is small, compared to that in the rock. In this case a pressure jump across the barrier can be taken into account. The power of such an approach is this capability to model simply a physical phenomenon while keeping the mathematical tools which already exist.

2. Single-phase flow in porous media with large faults

As an example we consider the model for incompressible single phase flow in a porous medium with a large fault.

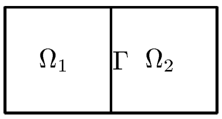

Figure 1. *A porous medium divided into two subdomains by a fault.*

A porous medium is split into two subdomains Ω_i $(i = 1, 2) \subset R^d$ $(d = 2$ or $3)$, by a fault Γ which is also a porous medium but with a different permeability (see Fig. 1). This fault is assumed to have a small width compared to the size of the whole domain, so it is modeled as a hypersurface $\Gamma \subset R^{d-1}$ between the subdomains. In the case when the permeability is greater in Γ than in the surrounding rocks, the flow is mainly driven into the fracture. When the permeability is smaller in the fault, Γ will play the role of a geological barrier disturbing the flow.

We are interested in studying the flow interactions between the rock and the fault. We assume that the flow in the subdomains as well as in the fault is governed by Darcy's law and that it is incompressible. For simplicity we neglect gravity. Thus we consider the following set of equations.

In the subdomains ($i = 1, 2$), mass conservation, Darcy's law, and the boundary conditions yield:

$$\begin{aligned}
\operatorname{div} \vec{\varphi}_i &= 0 && \text{in } \Omega_i, && [1]\\
\vec{\varphi}_i &= -K_i \vec{\nabla} p_i && \text{in } \Omega_i, && [2]\\
p_i &= p_f + (-1)^i \delta p && \text{on } \partial\Omega_i \cap \Gamma, && [3]\\
p_i &= p_d && \text{on } \partial\Omega_i \setminus \Gamma,
\end{aligned}$$

where K_i, p_i and $\vec{\varphi}_i$ denote the permeability, the pressure and the Darcy velocity in the subdomain Ω_i. Here p_d is the imposed Dirichlet boundary condition and $2\delta p$ $(= (p_2 - p_1)|_\Gamma)$ represents the jump of pressure across the fault.

In the fault:

$$\begin{aligned}
\operatorname{div}_f \varphi_f &= \vec{\varphi}_1 \cdot \vec{n}_1 + \vec{\varphi}_2 \cdot \vec{n}_2, && \text{on}\Gamma, && [4]\\
\varphi_f &= -\sigma_f \, K_f \vec{\nabla}_f \, p_f && \text{on } \Gamma, && [5]\\
4K_f \, \delta p &= \sigma_f (\vec{\varphi}_1 \cdot \vec{n}_1 - \vec{\varphi}_2 \cdot \vec{n}_2) && \text{on } \Gamma, && [6]\\
p_f &= p_{f,d} && \text{on } \partial\Gamma,
\end{aligned}$$

where K_f, σ_f, p_f and φ_f denote the permeability, the width, the pressure and the flow rate in the fault. The symbols $\vec{\nabla}_f$ and div_f are the gradient and the divergence operators along the fault. Note that the equations in the fault are written in dimension $(d - 1)$.

Equation [4] expresses mass conservation in the fault: the right-hand side represents the contribution from the subdomains into the fault. The subdomains are coupled with the fault through equation [6]. Pressure is not supposed to be continuous from Ω_1 to Ω_2, and it is possible to model the barrier case (when there is a low permeability fault). This is the main improvement in this model over the previous one (cf. [ALB 99]).

3. Approximation spaces

Let $\mathcal{O} \subset R^m$ ($m = 2$ or 3) be a polygonal domain. Then $\mathcal{O}$ is discretized with a structured or an unstructured mesh. Denote by $C \in \mathcal{T}$ a cell and by $E \in \mathcal{E}$ a face (if $m = 3$), or an edge (if $m = 2$) of the mesh. In this model, $\mathcal{O}$ will successively stand for the rock domains $\Omega_i \subset R^d$, $i = 1, 2$, and for the fault domain $\Gamma \subset R^{d-1}$.

In our framework, the scalar unknown, the pressure, as well as the vector unknown, the Darcy velocity, is approximated locally, cell per cell, by a polynomial. The approximations are discontinuous from one cell to the next. Inside each cell, for both the scalar and vector unknowns, there are cell degrees of freedom which are moments calculated over the cell, and edge degrees of freedom which are moments of the scalar function and moments of the normal components of the vector function over the faces. This is a generalization of standard finite volumes where scalars are approximated as piecewise constants, and where the vector unknowns have one degree of freedom per edge, the latter unknowns usually being eliminated.

However mathematical analysis as well as numerical experience show that the approximation spaces for the scalar and vector unknowns cannot be chosen arbitrarily. They must satisfy certain compatibility conditions which were studied in the analysis of mixed and mixed-hybrid finite elements. We need three approximation spaces.

1) The Darcy velocity $\vec{\varphi}$ is calculated in an approximation space $\vec{X}(\mathcal{O}) = \{\vec{v} \in (L^2(\mathcal{O}))^m \mid \vec{v}_{|C} \in \vec{X}_C, C \in \mathcal{T}\}$, where is $\vec{X}_C$ is a space of polynomials in C. Note that functions of $\vec{X}(\mathcal{O})$ are discontinuous from one cell to the next. However the fact that $\vec{X}_C$ is included in $H(\mathrm{div}, C)$ ensures that normal traces of functions in $\vec{X}_C$ on faces (or edges) can be defined.

2) The pressure p is first approximated inside the cells in a space $M(\mathcal{O})$ which is a subset of $L^2(\mathcal{O})$. Again functions of $M(\mathcal{O})$ are discontinuous from one cell to the next.

3) The pressure p is also approximated on the faces (or edges). We introduce TP which approximates the trace of the pressure on the faces (or edges) in the space $N(\mathcal{O}) \subset \prod_{C\in\mathcal{T}} \prod_{E\subset\partial C} L^2(E)$. Note that there are two traces per face (or edge) that we will denote by $(TP)^l$ and $(TP)^r$.

As already mentioned the spaces $\vec{X}(\mathcal{O})$, $M(\mathcal{O})$ and $N(\mathcal{O})$ cannot be chosen arbitrarily. Their choice must be based on the theory of mixed-finite elements and there is now a large catalogue of spaces available in two or three dimensions, for structured or nonstructured meshes [BRE 91, ROB 91]. Figure 2 gives examples in two dimensions.

4. Discretization of the flow equation

For the sake of clarity, let us assume that $d = 3$. We suppose that the complete domain $\bar{\Omega} = \bar{\Omega}_1 \cup \Gamma \cup \bar{\Omega}_2$ is first discretized, and then split. Therefore the mesh $\mathcal{T}(\Gamma)$ for Γ is the subset of the set of faces $\mathcal{E}(\Omega_i)$, $i = 1, 2$ of Ω_i, which are on the interface Γ. That is: $\mathcal{T}(\Gamma) \equiv \mathcal{E}_\Gamma(\Omega_i) \subset \mathcal{E}(\Omega_i)$, $i = 1, 2$.

Let us consider first the flow equation in the subdomains Ω_i, $i = 1, 2$, that we have written as a system of two equations, a conservation equation [1] and

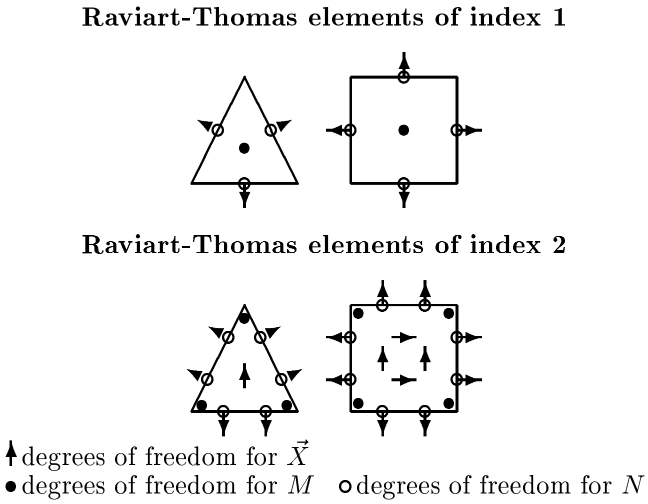

Figure 2. *Examples of mixed finite element spaces in two dimensions.*

Darcy's law [2]. To follow closely the physics we are going to write these two equations on each cell of the discretization and then equations on the edges expressing the intercell relations.

To simplify the notation we use the same notation for the approximation functions as for the continuous ones. So we would like to find $\vec{\varphi}_i \in \vec{X}(\Omega_i),\ p_i \in M(\Omega_i),\ TP_i \in N(\Omega_i)$ satisfying the following equations.

Cell equations :

1. *conservation equations*

$$\int_C \mathrm{div}\vec{\varphi}_i\, q = -\int_C \vec{\varphi}_i \cdot \vec{\nabla} q \ + \int_{\partial C} (\vec{\varphi}_i \cdot \vec{n}_C) q = 0, \quad q \in M(\Omega_i),\ C \in \mathcal{T}(\Omega_i). \quad [7]$$

This is equation [1], multiplied by a test function q and integrated over a cell C. It may or may not be integrated by parts. When it is, the degrees of freedom associated with the normal components appear explicitly in the approximation equation.

2. *Darcy's law*

$$\int_C K_i^{-1}\vec{\varphi}_i \cdot \vec{v} - \int_C p_i\, \mathrm{div}\vec{v} + \sum_{E\subset\partial C} \int_E TP_i\, \vec{v}\cdot\vec{n} = \int_C \rho\, \vec{q}_G \cdot \vec{v},$$

$$\vec{v} \in \vec{X}_C,\ C \in \mathcal{T}(\Omega_i). \qquad [8]$$

This equation is obtained by multiplying equation [2] by K_i^{-1} and by a test function $\vec{v}$ and then integrating over the cell C.

Edge equations, for edges within a subdomain:

3. *pressure continuity*

Since the pressure must be continuous across the edges, the two traces of the pressure must be equal, so $(TP)^l = (TP)^r$ on all interior edges. On the boundary, if we denote by $\mathcal{E}_{PD}(\Omega_i)$ the set of boundary edges where Dirichlet conditions are imposed on the pressure, we have also $TP_i = p_d$ on $E \in \mathcal{E}_{PD}(\Omega_i)$.

4. *conservation equations*

To preserve conservation the normal components of the total Darcy velocity must be continuous across the interior edges. If we assume also that we have homogeneous Neumann conditions on the part of the boundary where we do not have Dirichlet conditions, all these conditions can be written as $\int_E \vec{\varphi}_i^l \cdot \vec{n}\,\tau = \int_E \vec{\varphi}_i^r \cdot \vec{n}\,\tau, \quad \tau \in N(\Omega_i)$, where $\vec{n}$ denotes one of the unit normals to E.

Edge equations, for edges between a subdomain and the fault:

3bis. *pressure discontinuities*

Since traces on the edges are available in our approximation framework, equations [3] and [6] can be approximated in a straightforward manner. On the fault interface, if we denote by $\mathcal{E}_\Gamma(\Omega_i)$ the set of edges on the fault, we have:

$$TP_1 = P_f - \delta p \quad \text{and} \quad TP_2 = P_f + \delta p \quad \text{on } E \in \mathcal{E}_\Gamma(\Omega_i) = \mathcal{T}(\Gamma), \qquad [9]$$

$$4K_f\,\delta p = \sigma_f(\vec{\varphi}_1 \cdot \vec{n}_1 - \vec{\varphi}_2 \cdot \vec{n}_2) \quad \text{on } E \in \mathcal{E}_\Gamma(\Omega_i) = \mathcal{T}(\Gamma). \qquad [10]$$

Equations along the fault:

One can simply discretize the equations along the fault in the same way as in the subdomains Ω_i, but this time in $(d-1)$ dimension.Thus the conservation equation [4] and Darcy's law [5] along the fault may be written:

$$\int_C \mathrm{div}\vec{\varphi}_f\, q - \int_C (\vec{\varphi}_1 \cdot \vec{n}_1 + \vec{\varphi}_2 \cdot \vec{n}_2)\, q = 0 \quad q \in M,\, C \in \mathcal{T}(\Gamma),$$

$$\int_C (K_f)^{-1}\vec{\varphi}\cdot\vec{v} - \int_C p_f\, \mathrm{div}\vec{v} + \sum_{E\subset\partial C} \int_E TP_f\, \vec{v}\cdot\vec{n} = \int_C \rho\,\vec{q}_G\cdot\vec{v}, \quad \vec{v} \in \vec{X}_C,\, C \in \mathcal{T}(\Gamma)$$

We call the above method a finite volume method for the following reasons:

– Equations have been written cell by cell and intercell relations are expressed by edge equations.

– The scheme is cellwise conservative. Indeed since the approximation space M contains piecewise constants we have

$$\int_C \mathrm{div}\vec{\varphi} = \sum_{E\subset\partial C} \int_E \vec{\varphi}\cdot\vec{n}_C = 0,\ C \in \mathcal{T}.$$

– Let us consider the case in which the domain is discretized with rectangular meshes with faces (or sides) parallel to the coordinate axes and the Raviart-Thomas-Nedelec elements of lowest index are used. If one uses the trapezoidal rule in all the coordinate directions to calculate the integral $\int_C (Kd)^{-1}\vec{\varphi}\cdot\vec{v}$ one can eliminate the total velocity $\vec{\varphi}$ and obtain the standard cell-centered finite volume method with a five-point stencil in two dimensions or a seven-point stencil in three dimensions [JAF 84, RUS 84, WEI 88]. On triangular meshes the relation between mixed finite elements and cell-centered finite volumes has been investigated in [AGO 95].

One should also note that the above method is equivalent to the mixed or mixed-hybrid finite element method in the sense that the numbers for p and $\vec{\varphi}$ obtained from the machine are the same for both methods. The difference in the formulation lies in the fact that in our finite volume method all continuity requirements are enforced explicitly.

Once the approximate formulation has been written, the resulting system of equations is very large and the next step is to choose the most efficient way to implement it. Various possibilities exist but this is beyond the scope of this paper.

5. Numerical results

We show in figure (3) the pressure field when the permeability in the fault is 200 times larger than in the surrounding rock. The boundary conditions are: a pressure drop from 1 to 0 from left to right, and the same pressure drop from 1 to 0 in the fault from top to bottom; elsewhere, homogeneous Neumann boundary conditions. In figure (3) we observe that the pressure is continuous across the fault.

On the contrary, in figure (4), the permeability is 200 times smaller in the fault than in the rest of the domain. This time, we observe that the barrier causes a pressure drop between the two subdomains: that pressure is no longer continuous across the fault.

6. Conclusion

We have presented a general framework for finite volume methods for structured as well as for nonstructured meshes, which allows for the use of non-constant approximation inside the discretization cells. These methods use the approximation spaces provided by mixed-hybrid finite element methods. Using incompressible two-phase flow as an example, it was shown that this approximation technique is very convenient for problems with heterogeneities, and that even in the homogeneous case, it may lead to new schemes.

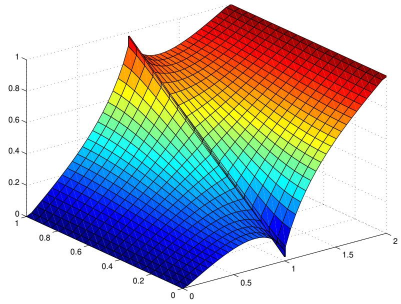

Figure 3. *Pressure field with a high permeability fracture.*

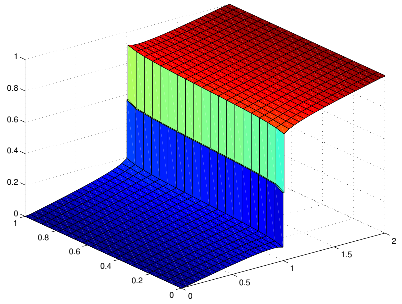

Figure 4. *Pressure field when a geological barrier separates the rock.*

References

[ALB 99] C. Alboin, J. Jaffré, J.E. Roberts, X. Wang, C. Serres. Domain decomposition for some transmission problems in flow in porous media. In Z. Chen, R.E. Ewing, Z.-C. Shi editors, *Numerical treatment of multiphase flows in porous media*, 22–34, Springer, Berlin, 1999.

[AGO 95] A. Agouzal, J. Baranger, J.-F. Maître, F. Oudin. Connection between finite volumes and mixed finite element methods for a diffusion problem with nonconstant coefficients, with application to convection-diffusion. *East-West Journal on Numerical Analysis*, 3:237–254, 1995.

[BRE 91] F. Brezzi, M. Fortin. *Mixed and Hybrid Finite Element Methods.* Springer Verlag, Berlin, 1991.

[JAF 84] J. Jaffré. Décentrage et éléments finis mixtes pour les équations de diffusion-convection. *Calcolo*, 21:171–197, 1984.

[ROB 91] J.E. Roberts, J.-M. Thomas. Mixed and hybrid methods. In P.G. Ciarlet and J.L. Lions, editors, *Handbook of Numerical Analysis Vol.II*, 523–639. North Holland, Amsterdam, 1991.

[RUS 84] T. Russell, M.F. Wheeler Finite element and finite difference methods for continuous flows in porous media. in R. E. Ewing ed., *The Mathematics of Reservoir Simulation*, SIAM, Philadelphia, 1984, 35–106.

[WEI 88] A. Weiser, M.F. Wheeler. On convergence of block-centered finite differences for elliptic problems. *SIAM J. Numer. Anal.*, 25:351–375, 1988.