Abstract

The generation of accurate and reliable unstructured 3D models for reservoir simulation remains a challenge. In this paper, new developments for grid generation, upscaling and streamline simulation for such models are described. In combination, these techniques provide a prototype workflow for the construction of unstructured simulation models. The grid generation framework described here allows for the incorporation of both geometrical constraints and grid-resolution targets. Flow adaptation of the unstructured grid (i.e., higher grid density in key regions) is accomplished through the use of single-phase flow calculations on the underlying geocellular grid, which are used to generate target grid resolution maps for the unstructured coarse model. A novel transmissibility upscaling procedure is introduced to capture the effects of fine scale heterogeneity. A new method for streamline simulation on unstructured grids is also introduced. This technique provides an efficient flow-based diagnostic for the assessment of the coarse simulation model in terms of flow response. The performance of the various components of the methodology is demonstrated through application to several example cases.

1. Introduction

The recent evolution of reservoir simulation is toward more structurally complex geological models and increasingly detailed petrophysical property descriptions. In order to manage reservoir uncertainties, reservoir simulation studies may now also require multiple models to be generated, often with different geological scenarios. It is a key challenge to generate gridded reservoir descriptions that incorporate the structural complexity of the geology while maintaining model sizes that are practical for flow simulations. A general approach for addressing the high level of detail in geocellular models is to upscale; i.e., introduce a significant degree of grid coarsening. Flexible (unstructured) grids provide an attractive solution for this coarsening step, as they enable the accurate and efficient modeling of both the reservoir geometry and heterogeneity.

Several issues arise with unstructured grids, particularly within the context of grid generation and upscaling. The data structure must enable the construction of grids that conform to geometrically complex 3D surfaces and honor complex surface intersections and topology constraints. This issue is handled by a sophisticated gridding framework called a Soft Frame Model. In this work, grids are generated to conform not only to geological features but are also used to introduce higher resolution in regions of high flow. This is accomplished by solving a representative flow problem on the fine scale and using the flow information in the grid construction step. Upscaled properties must then be computed for these unstructured flow-based grids. This work proposes a novel approach for this upscaling problem. Finally, a grid quality assessment based on streamline simulations for tracer flow is proposed. For this aspect of the problem, we generalize streamline simulation techniques to unstructured grids.

The outline of this paper is as follows. We first present the basic issues and workflow for reservoir simulation with unstructured grids. Next we describe the techniques used for grid generation (both in terms of structure and flow resolution). Then, we discuss the generation of the coarse model, specifically the upscaling procedure, and the development of a streamline-based simulator capable of obtaining flow responses on (unstructured) coarse models. Finally, we illustrate the overall methodology on several example cases.
2. Problem overview

Our starting point is a fine scale geocellular model, which may be defined on an unstructured or a structured grid, as well as a description of the key structural geological features. Our overall procedure is then as follows. For a particular set (or sets) of boundary or well conditions, we perform a fine scale steady-state single-phase flow simulation. This calculation is generally not overly expensive because it need only be performed once. From this solution, we generate streamlines and simulate the behavior of an incompressible tracer flow by tracking particles along streamlines. This is a very inexpensive calculation for structured models. Next, using the flow information as well as structural information, we generate a coarse scale unstructured tetrahedral grid. This represents the primal grid. The dual grid is constructed by forming control volumes that are centered at the nodes (vertices) of the primal grid.

We upscale the underlying fine scale geocellular description to the scale of the control volumes. This provides the coarse scale model. We discretize the governing equations on the dual (coarse) grid using a control volume finite element (CVFE) method (Verma and Aziz, 1997). In this method, fluxes across control volume faces are represented in terms of the cell-centered pressures. These fluxes may be defined in terms of the pressures for only the two cells sharing the interface (in this case we have a two-point flux approximation or TPFA) or in terms of these cell pressures as well as those of other neighboring cells (multipoint flux approximation or MPFA). Next, we perform the same steady-state single-phase flow simulation on the coarse grid as was run on the fine scale. This allows us to compare the flow responses of the two models and to iterate on the coarse grid structure if necessary. The tracer solution on the coarse scale is accomplished using a streamline simulator developed especially for CVFE solutions on unstructured grids.

The overall workflow described above for the construction of unstructured simulation models is illustrated schematically in Figure 1. There are several new elements in this methodology, including the unstructured grid generation, the procedures for transmissibility upscaling and streamline simulation on unstructured grids, and the linkage of the fine scale flow simulation to the grid generation procedure. We now consider each of these issues in turn.

3. Grid generation methodology

The proposed methodology relies on a flow-based, quality controlled grid construction. It starts from a structural model that must first be “enriched” to ensure that it provides a valid representation of the topological constraints. Then, appropriate gridding algorithms operate on the initial grid and adapt the cell shape and size to user-defined resolution constraints while maintaining the topological and geometrical validity of the grid. This construction relies on a topological structure that embeds the concepts of a grid structural model as well as grid topology and resolution constraints.

This section briefly describes the key aspects of the gridding framework. First, the data structures developed for the grid generation methodology are considered, followed by a discussion of the grid generation algorithms. Finally, we describe how the flow response from the fine scale property model is incorporated into the grid generation. Note that the grid generation may be applied as the first step of the overall procedure (i.e., to define the fine scale model) or it may not be used until the coarse grid generation step.

3.1 Soft Frame Model

From a geometrical point of view, a structural model is a set of discrete 3D representations of geological objects such as faults, horizons, and their borders. Objects of dimension $p$ ($p$ in $[0,2]$) are represented by grids comprised of elements of dimension $p$ called “$p$-cellular complexes”. These objects and their borders have various topologies (e.g., they can be either closed or open). Building such structural models is a crucial research topic and numerous techniques have been proposed (see e.g., Caumon et al., 2004). In the following, we will consider the structural model to be an input.
In general, every p-cellular complex is built independently; i.e., surfaces representing faults and horizons do not share any borders (even though they may physically intersect). In order to construct a volumetric mesh that accounts for the potentially very complex contacts between the geological objects, we require a “valid” topological and geometrical representation of the contacts. This is achieved using a macro model called a Soft Frame Model (Lepage, 2003). With this representation, a contact of dimension $p$ (called a $p$-radial element) is defined as the association of $p+1$ intersecting objects plus their $p$ intersections. Based on that definition, relative incidency and adjacency relationships are defined. These allow us to represent any structural model with an incidency graph (Figure 2). The ensemble of all radial elements forms a Soft Frame Model. This representation enables us to manipulate the contacts efficiently and flexibly. Furthermore, the geometrical consistency (every contact has a unique geometry shared by all the intersecting objects that define it) is ensured by generating meshes for all the radial elements of the Soft Frame Model, incorporating known sets of geometrical constraints such as nodes, edges or faces. This approach is conceptually very different from the standard proposed solutions based on the computation of the geometrical intersections between objects (see e.g., Caumon et al., 2004).

3.2 Grid Refinement

Within our gridding framework, meshes are constructed as a series of constrained cellular complexes, from lower to higher dimensions (points, lines, surfaces and volumes), where lower dimensional objects serve as constraints for higher dimensional objects. To honor constraints in the triangulation, we apply a conforming technique, which tends to produce high-quality triangles in terms of shape. This is the case because the resulting triangles are all strongly Delaunay. The process of honoring constraints in the triangulation may require the insertion of extra points (Steiner points) in the mesh, in contrast to boundary-constrained Delaunay meshes, where the circumcircle of some of the triangles may contain other points. A key problem with these insertions is the possible lack of convergence of the gridding algorithm when the constraining elements bear small angles. Our technique successfully addresses this issue by applying appropriate bisection rules expressed in the low-distorted parameterized spaces where triangulations are built.

Ensuring a satisfying shape for a tetrahedron is more difficult than for a triangle. Indeed, it is impossible to build a well-shaped tetrahedron based on a quasi-degenerated face. As described before, a conforming approach may be used to honor constraints in the tetrahedralization. However, it would produce far too many Steiner points in practice (Shewchuck, 2002), which is in contradiction to our objective. A standard boundary-constrained Delaunay methodology was therefore applied. It uses finite sequences of edge and face swaps plus a point insertion mechanism to recover constraints in the mesh.

To optimize the shape of the resulting tetrahedra, we propose an approach with no insertion of Steiner points. This produces tetrahedra with a bounded circumradius-to-shortest-edge ratio in a portion of the mesh, while keeping the boundary-constrained Delaunay property. Badly shaped tetrahedra are removed from the mesh by inserting their circumcenter into the tetrahedralization, as in any Delaunay refinement technique.

3.3 Flow Adaptation

Using the Soft Frame Model and gridding algorithms presented above, we generate flow-adapted grids according to the following sequence. We perform incompressible tracer flow simulations on the reference geocellular model for one or more sets of boundary conditions. While the flow responses thus obtained serve as a reference for the diagnostic that controls the iterations of our overall methodology (Figure 1), the overall solution is used to generate a velocity map for each flow problem. Then, with appropriate scalings, this information is averaged into a mean flow rate map that is then transformed into a target resolution constraint. The appropriate use of the flow information is the real challenge of the grid adaptation. This issue is addressed here by defining gridding parameters and investigating the search space in order to find optimum combinations that minimize the number of simulation cells while preserving the flow response. These gridding parameters include the aspect ratio (or anisotropy) of the tetrahedral grid elements. The layered aspect of petrophysical property distributions suggests the use of elongated tetrahedra. This can be achieved by computing element sizes using a metric (rather than plain
Euclidean distance) when searching for elements that violate the resolution constraint. The size ratio between small elements (used in high flow regions) and large elements is another important grid parameter. The sensitivity of simulation results to several of the key gridding parameters is reported in Prévost (2003).

4. Transmissibility upscaling

Upscaled properties must be calculated for the coarse grid. The dual cells of the flow-adapted grid (obtained as described above) are the control volumes used in the CVFE discretization. The objective of the upscaling step is to provide the flow simulator with (dual) cell to cell transmissibility coefficients that relate flow across an interface to values of the pressure at the center of neighboring (dual) cells. These coefficients must capture the flow effects of the underlying fine scale permeability description. The transmissibilities in the upscaled model can be calculated in at least two different ways. The first method, referred to as a $k^*$-MPFA upscaling, entails the calculation of an upscaled permeability tensor for each control volume and the subsequent determination of the multipoint flux CVFE stencil from the $k^*$ values in neighboring cells. The second method, referred to as a $T^*$-TPFA upscaling, entails a direct computation of the two-point transmissibility coefficients between pairs of connected cells. Both of these upscaling procedures were implemented and evaluated (Prévost, 2003). Numerical tests showed that the $T^*$-TPFA method is generally more robust and accurate for the types of problems tested. We now describe the $T^*$ calculation in more detail.

Consider an interior face of the dual grid and the two corresponding adjacent cells (as shown in Figure 3). The two cells are polyhedra and the face is a polygonal surface which, in the case of a CVFE dual grid, is in general nonplanar. In TPFA, the flux across a control volume face is written in terms of the difference between pressure values at the centers of the dual cells. Denoting $ijF$ as the face between cells $i$ and $j$, the flux $ijq$ across $ijF$ can be written as:

$$-( -  )_{ij} = T_{ij} (P_j - P_i),$$

(1)

where $T_{ij}$ is the two-point transmissibility coefficient.

We compute $T_{ij}$ by assembling and solving a local flow problem, as illustrated in Figure 4. The local domain includes the two control volumes that share the target interface as well as a small border region around the two cells. We solve the single-phase pressure equation subject to pressure – no flow boundary conditions over this fine scale region and then compute average pressures over cells $i$ and $j$, designated $\langle P_i \rangle$ and $\langle P_j \rangle$. The average Darcy velocity can also be calculated over fine scale cells contained in the interface $ijF$, allowing the determination of the total flux $ijq$ across $ijF$. The upscaled transmissibility is then calculated via:

$$T^*_{ij} = \frac{q_{ij}}{\langle P_i \rangle - \langle P_j \rangle}.$$

(2)

We note that this transmissibility upscaling procedure requires that a local problem be solved for each connection (interface) in the domain, which can be somewhat time consuming.

5. Streamline simulation on unstructured grids

As indicated earlier, we use a simplified flow model (incompressible tracer flow) to gauge the accuracy of the coarsened reservoir description. This model will of course not correspond exactly to the actual physical situation of interest in most cases, but it represents a reasonable approximation for purposes of coarse model evaluation. We apply streamline simulation for this calculation, as it is one of the most efficient techniques for simulating incompressible tracer flow. If the boundary or well conditions do not change during the course of the simulation, the complete production history can be obtained with only
one solution of the pressure equation. The producing fraction of the injected tracer is computed by tracking a number of streamlines from injectors to producers and plotting, for each producer, the streamline time of arrival (sorted in increasing order) versus the cumulative sum of the rates associated with the streamlines. For simple one-producer one-injector cases, the time axis of the production curves can easily be expressed in terms of pore volume injected.

The quality of the coarse model is assessed based on the quantitative level of agreement between the flow-response obtained on the fine and coarse grids for one or more flow problems. In most cases, we use tracer flow simulations from one face of the reservoir to the other (under fixed pressure boundary conditions) and compare total flow rates across the model and nondimensional fractional flow curves (i.e., water cut). For that purpose, a streamline-based simulator for 3D unstructured grids was developed. We note that the streamline simulator allows for the use of any set of boundary or well conditions.

The principal difficulty in the extension of the streamline method to unstructured grids lies in the determination of an appropriate analytical velocity interpolant, which is required for the tracing. Our technique consists of a local postprocessing of the numerically calculated fluxes, leading to a consistent and flux-continuous piecewise constant representation for the velocity. Fluxes are obtained from the pressure field computed using the CVFE discretization scheme. For each control volume, the postprocessing introduces a subgrid comprised of tetrahedra, each of which is associated with an unknown velocity vector. The set of velocity vectors are then constrained to satisfy certain physically meaningful constraints: flux-continuity, consistency with the fluxes derived from the CVFE solution, and minimization of the rotational component of velocity. The local constraints are expressed as a linear system of equations which scales as the number of tetrahedra shared by the control volume. Solution of this set of equations (for each control volume) provides a velocity field that can be used for streamline tracing. See Prévost (2003) for full details.

As an illustration of the performance of the unstructured streamline simulation method, we show a result for which comparison to a structured grid streamline simulation (Batycky et al., 1997) is possible. We consider a homogeneous cubic domain and simulate flow from one corner to the diagonally opposite corner. For the unstructured simulations, a total of 1728 control volumes were used; for the structured simulations a total of 27,000 cells were used (though similar results could be obtained with fewer cells). Results for water cut ($F_w$) versus PVI are shown in Figure 5, the unstructured grid used is shown in Figure 6. The excellent agreement demonstrates the accuracy of our unstructured streamline simulations.

### 6. Results for unstructured grid generation and upscaling

We now present examples demonstrating the capabilities of the general grid generation procedure as well as flow results illustrating the flow-based gridding and upscaling techniques. In all cases, a Soft Frame Model was built from an initial structural model, surfaces were remeshed using a conforming approach, volumes were tetrahedralized, optimized according to a shape criterion, and post-processed through an edge-swap technique.

Figure 7 gives an illustration of the meshes produced by our methodology. The structural model is comprised of top and bottom horizons intersected by a complex network of faults. In addition to these surfaces, the faces of a bounding box are included in the set of constraints. The gridded model exhibits an accurate resolution of surface intersections and a very acceptable shape for the tetrahedra. Figure 8 shows another example that demonstrates the performance of the grid refinement algorithms in the context of surfaces intersecting at low angles.

Figure 9 shows an example of the combination of structural and flow adaptation. The fault geometry is honored in both the primal and the dual grid. The resolution constraint is provided by a mean flow rate map calculated on the fine model and averaged for different flow problems. The dual grid displays the norm of the upscaled permeability tensor. The unstructured coarse model can now be used in flow simulations.

Finally, we demonstrate the benefit of flow-adaptation using the methodology presented. We consider a fine 200×100×50 Cartesian model with a layered, log-normally distributed permeability field of variance...
(in log $k$) of 1 and dimensionless correlation lengths $l_x=1.0$, $l_y=0.75$ and $l_z=0.2$ (shown in Figure 10). The flow adaptation was performed in three steps: (1) determine the optimum target aspect ratio from uniformly coarsened grids, (2) determine the optimum large-to-small cell size ratio, (3) select the grid size (number of cells) to achieve the desired level of accuracy (based on the flow diagnostics).

The optimum aspect ratio is defined as the aspect ratio that provides the most accurate coarse models (relative to the reference solution). For this case it was found to be 10×5×1 (i.e., a well shaped tetrahedron would fit in an ellipsoid with principal axis lengths in the ratios 10:5:1). Then, enforcing this optimal aspect ratio for the tetrahedral cells, grid adaptation was performed. This required the determination of the optimum large-to-small tetrahedra size ratio and the selection of the desired number of coarse cells. The performance of the resulting “optimum” coarse model (relative to the fine model) is shown in Figure 11. Here, $Q_c$ is the ratio of the total flow rate through the coarse model to that of the fine model ($Q_c=1$ indicates exact agreement). Using 1394 control volumes, the flow-rate adapted grid provides an error in total flow of less than 1% and very accurate results for water cut ($F_w$) relative to the fine (10^6 cells) model. The figure also shows results using a uniformly coarsened model with nearly twice as many cells as the flow-rate adapted grid. The errors in both flow rate and water cut are considerably higher with this model, demonstrating the benefits of the flow-rate adaptivity applied here.

**Figures**

![Figure 1: Schematic of overall workflow for the generation of an unstructured simulation model.](image1)

![Figure 2: Soft Frame Model representation using an incidence graph. The model is comprised of three radial surfaces, seven radial lines and seven radial nodes. For any radial element, the graph gives the set of geometrical constraints to honor when building a mesh.](image2)
Figure 3: $k^*$ is calculated on 3D dual cells $i$ and $j$, while two-point transmissibility is calculated through the $F_{ij}$ face.

Figure 4: Two-point transmissibility is calculated by performing a local single-phase flow simulation on a local rotated grid.

Figure 5: Comparison of unstructured streamline simulation results to those from a structured streamline simulator.

Figure 6: Grid used for the unstructured streamline simulation (wells located at opposite corners).

Figure 7: Example of a complex structural model (left) and corresponding tetrahedralization (Nancy IV model courtesy of LIAD).
Figure 8: Exploded view of a gridded model containing a “Y” fault. Note that the small angles appearing at the edges of the interior ‘lens’ are accurately resolved.

Figure 9: The adapted grid (lower left) is constructed from structural constraints (top left) and resolution constraint (lower right) obtained from flow rates calculated on the permeability model (top right). Note the variable grid density and the dual cells that honor the fault geometry.

Figure 10: Using a user defined grid anisotropy and a resolution constraint derived from a mean flow-rate map, an adapted coarse model is constructed. The property displayed on the upscaled grid is the norm of k*= (log scale).

Figure 11: Fractional flow curves for tracer simulation for flow from left to right. Both total flow rate (Qc) and water cut for the coarse model are considerably improved by adaptation to flow rate information.

References


