DOE/BC/15213-3
(OSTI ID: 784657)

ADVANCED TECHNIQUES FOR RESERVOIR SIMULATION AND
MODELING OF NON-CONVENTIONAL WELLS

Annual Report
September 1, 2000-August 31, 2001

By:
Louis J. Durlofsky
Khalid Aziz

Date Published: August 2001

Work Performed Under Contract No. DE-AC26-99BC15213

Stanford University
Stanford, California

**National Energy Technology Laboratory
National Petroleum Technology Office
U.S. DEPARTMENT OF ENERGY
Tulsa, Oklahoma**

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, expressed or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government.

This report has been reproduced directly from the best available copy.

Advanced Techniques for Reservoir Simulation and Modeling of Non-Conventional Wells

By
Louis J. Durlowsky
Khalid Aziz

August 2001

Work Performed Under DE-AC26-99BC15213

Prepared for
U.S. Department of Energy
Assistant Secretary for Fossil Energy

Tom Reid, Project Manager
National Petroleum Technology Office
P.O. Box 3628
Tulsa, OK 74101

Prepared by
Department of Petroleum Engineering
School of Earth Sciences
Stanford University
Stanford, CA 94305-2220

Contents

1. General Introduction and Overview	1
Part I. Development of Advanced Reservoir Simulation Techniques for Modeling Non-Conventional Wells	5
2. Development of a General Purpose Research Simulator	5
2.1 Introduction	5
2.2 General Formulation of GPRS	7
2.3 Preliminary Results	19
2.4 Preliminary Conclusions and Future Directions	31
2.5 References	35
3. Flow-Based Modules for Grid Generation in Two and Three Dimensions	37
3.1 Introduction	37
3.2 Modular versus Global Grid Generation	38
3.3 K-Orthogonal Streamline-Equipotential Grids	39
3.4 Generation of Flow Response Based Modules	41
3.5 Upscaling and Grid Smoothing	44
3.6 Flow Simulation Results	45
3.7 Conclusions	53
3.8 References	53
Part II. Coupling of the Reservoir and Non-Conventional Wells in Simulators	55
4. Investigation of Drift-Flux Model Parameters	55
4.1 Introduction	55
4.2 Basic Definitions for Two Phase Pipe Flow	57
4.3 Drift-Flux Model Parameters	58
4.4 Drift-Flux Model in Different Flow Patterns	60
4.5 Method for Parameter Determination	61
4.5.1 Objective Function Using α_G	61
4.5.2 Incorporation of α_G into Correlations of C_0 and V_d	68
4.6 Application of Proposed Method to Other Inclination Angles	71
4.7 Discussion of Drift Velocity V_d	76
4.7.1 Physical Meaning of C_0 and V_d	76
4.7.2 Further Investigation of V_d as $\alpha_G \rightarrow 1$	78
4.7.3 Effects of Inclination Angles on V_d	80
4.8 Evaluation of the Drift-Flux Model in Eclipse	82
4.9 Conclusions and Recommendations for Future Work	87
4.10 References	90
Part III. Novel Approaches to Account for Heterogeneities in the Vicinity of Non-Conventional Wells	93

5. Semi-Analytical Modeling of Complex Wells with Downhole Inflow Control Devices.....	93
5.1 Introduction	93
5.2 Modeling of Reservoir Inflow	96
5.3 Modeling of Downhole Pressure Losses.....	98
5.4 Residual and Jacobian Construction	99
5.5 Application to Complex Reservoirs and Wells.....	102
5.5.1 Validation of the Overall Implementation	102
5.5.2 Single Heterogeneous Reservoir	102
5.5.3 Multiple Reservoirs	108
5.5.4 Dual-lateral Well	112
5.6 Conclusions	115
5.7 References	118
 6. Numerical Calculation of Equivalent Cell Permeability Tensors for General Quadrilateral Control Volumes	 120
6.1 Introduction	120
6.2 Governing Equations and Numerical Procedures	123
6.2.1 Pressure Equation and Periodic Boundary Conditions	124
6.2.2 Finite Element Solution Procedure	126
6.2.3 Treatment of Geometric Effects: Quadrilateral Cells	127
6.2.4 Alternative Approaches, Treatment of Polygonal Cells, and Gridding.....	131
6.3 Numerical Results Using the Conforming Scale Up Method	133
6.4 Discussion and Conclusions.....	141
6.5 References	143

Abstract

Research results for the second year of a five year project on the development of improved modeling techniques for non-conventional (e.g., horizontal, deviated or multilateral) wells are presented. The overall program entails the development of enhanced well modeling and general simulation capabilities. A general formulation for black-oil and compositional reservoir simulation is presented. This formulation is very flexible in terms of variable selection and the numerical treatment of unknowns. Results demonstrating reduced computation times for a compositional problem are presented. A procedure for generating flow-based grids for two and three dimensional problems is described. This technique acts to concentrate grid lines in important high flow regions. The applicability of the method for coarse scale simulation is illustrated through several example cases. An accurate model for two-phase wellbore flow is also developed. This model, of the drift-flux type, is suitable for use within a reservoir simulator. Improved representation of experimental data, relative to existing models, is demonstrated. New techniques for the efficient representation of permeability heterogeneity are described. These include a semi-analytical approach for modeling non-conventional wells that approximates the effects of reservoir heterogeneity, wellbore hydraulics and downhole inflow control devices. A procedure for upscaling fine grid geostatistical models to general quadrilateral simulation cells is also presented. Application of these techniques for heterogeneity modeling is shown to provide accurate and efficient flow models.

Executive Summary

Non-conventional wells, which include horizontal, deviated, multilateral and “smart” wells, are essential for the efficient management of oil and gas reservoirs. These wells are able to contact larger regions of the reservoir than vertical wells and can also be used to target isolated hydrocarbon accumulations. Because non-conventional wells can be very expensive to drill and complete, it is important to be able to accurately model their performance. However, predictions of non-conventional well performance are often inaccurate. This is likely due in part to inadequacies in many of the reservoir engineering and reservoir simulation tools used to model non-conventional wells.

A number of complications appear in the modeling of non-conventional wells that do not exist, or exist to a much lesser extent, with vertical wells. Because non-conventional wells introduce additional geometric complexity into the system, more flexible procedures are required for gridding and discretization in models that include these wells. In addition, the effects of wellbore hydraulics, which are generally neglected for vertical wells, can be very significant for long horizontal or deviated wells. Because these wells often operate under very low drawdowns, pressure losses in the wellbore can have an adverse effect on productivity. Therefore, efficient models for wellbore flow must be incorporated into the overall reservoir simulation. The effects of near-well heterogeneity, in addition to key geologic features away from wells, can also have a significant impact on non-conventional well productivity. It is therefore important to include these effects in flow models of non-conventional wells. Finally, because it is essential that the tools developed to address each of these individual issues be integrated, it is necessary to develop a general simulation capability that is able to accommodate all of these new modeling procedures.

Our research addresses all of the areas discussed above. The work is divided into three main categories: (1) advanced reservoir simulation techniques for modeling non-conventional wells; (2) improved techniques for computing well productivity (for use in reservoir engineering calculations) and well index (for use in simulation models); and (3) accurate approaches to account for the effects of reservoir heterogeneity. Our progress during the second year of funding in each of these main areas is as follows.

A general purpose object-oriented research simulator is currently under development. This simulator will eventually integrate many of the new tools developed for the improved modeling of non-conventional wells. This year, we completed the development of general black-oil and compositional modeling modules in the simulator. The formulation is very general in that it allows for the selection of a variety of primary and secondary variables and accommodates varying degrees of implicitness. For example, in a compositional setting, the options in existing simulators include IMPEC (implicit in pressure and explicit in all compositions) and fully implicit (implicit in all variables). In our new formulation, an additional option includes an intermediate level of implicitness in which both pressure and saturation are implicit but all other variables are explicit. This new formulation is shown to provide accurate and efficient results in test cases.

We are also developing general grid generation procedures for use in reservoir simulation. In this report we describe our recent work on the generation of flow-based grids. These grids use streamlines and isopotentials generated from a simplified flow problem to define a coarsened reservoir grid, which is then used for two phase flow simulations. Significantly improved results, relative to those obtained using standard procedures, are demonstrated for two and three dimensional flow problems. The grid generation can be accomplished either globally (over the entire reservoir model) or modularly (over only a limited region of the model).

As indicated above, non-conventional well productivity can be impacted by wellbore hydraulics. An important aspect of the wellbore flow model is the accurate determination of the in situ volume fractions (i.e., liquid holdup) within the wellbore. We present improved models for two phase gas-liquid pipe flow systems. These models, which relate gas velocity and in situ gas fraction to the mixture velocity, provide improved correlations over those of existing models. The new parameterizations were developed using the large amount of experimental data contained within the Stanford multiphase flow database.

Reservoir heterogeneity is another important effect that must be captured in flow models of non-conventional wells. Here we present two investigations aimed at better representing permeability variations. The first technique described is a semi-analytical approach that utilizes Green's functions. In this formulation the wellbore itself is discretized rather than the reservoir (as in a finite difference method). This technique is highly efficient although it is limited to single phase flow problems. The methodology includes approximate models for the effects of heterogeneity, wellbore hydraulics and downhole inflow control devices such as chokes. Thus, the method represents an important component in the modeling of "smart" wells. Results are presented for a variety of systems and demonstrate the applicability of the approach for optimizing production with non-conventional wells operating with downhole inflow control devices.

We next present a general approach, suitable for use with finite difference flow models, for computing upscaled cell permeability tensors for general quadrilateral (i.e., non-orthogonal) cells. This method applies an underlying finite element technique to accurately account for both fine scale heterogeneity and the coarse cell geometry. The accuracy of the new method is demonstrated for several examples including some with random grids. The real benefit of the method is evident when it is used in conjunction with general gridding techniques. We therefore apply the method to a two dimensional grid generated using a flow-based technique, as described above. In this case significantly improved results, over those achieved using standard techniques, are obtained.

Papers from this and earlier work:

- Aziz, K. and Ouyang, L.-B.: "Productivity of Horizontal and Multilateral Wells," *Pet. Sci. and Tech.*, **19**, 1007-1023, 2001.
- Castellini, A., Edwards, M.G. and Durlofsky, L.J.: "Flow Based Modules for Grid Generation in Two and Three Dimensions," Proceedings of the 7th European Conference on the Mathematics of Oil Recovery, Baveno, Italy, Sept. 5-8, 2000.
- Edwards, M.G.: "Split Full Tensor Discretization Operators for Structured and Unstructured Grids in Three Dimensions," SPE paper 66358, presented at the SPE Reservoir Simulation Symposium, Houston, Feb. 11-14, 2001.
- He, C., Edwards, M.G. and Durlofsky, L.J.: "Numerical Calculation of Equivalent Cell Permeability Tensors for General Quadrilateral Control Volumes" (in review for *Computational Geosciences*).
- Ouyang, L.-B. and Aziz, K.: "Transient Gas-Liquid Two-Phase Flow in Pipes with Radial Influx or Outflux," *J. Pet. Sci. and Eng.* (in press, 2001).
- Petalas, N. and Aziz, K.: "A Mechanistic Model for Multiphase Flow in Pipes," *J. Can. Pet. Tech.*, **39**, 43-55, 2000.
- Prevost, M., Edwards, M.G. and Blunt, M.J.: "Streamline Tracing on Curvilinear Structured and Unstructured Grids," SPE paper 66347, presented at the SPE Reservoir Simulation Symposium, Houston, Feb. 11-14, 2001.
- Valvatne, P.H., Durlofsky, L.J. and Aziz, K.: "Semi-Analytical Modeling of the Performance of Intelligent Well Completions," SPE paper 66368, presented at the SPE Reservoir Simulation Symposium, Houston, Feb. 11-14, 2001.
- Wan, J., Penmatcha, V.R., Arbabi, S. and Aziz, K.: "Effects of Grid Systems on Predicting Horizontal Well Productivity," *SPE J*, **5**, 309-314, 2000.

Presentations on this work were given at the following conferences and meetings:

- SPE Annual Technical Conference and Exhibition, Dallas, TX, Oct. 2000
- SPE Reservoir Simulation Symposium, Houston, TX, Feb. 2001 (three presentations)
- SPE Angola Section, Luanda, Angola, March 2001
- Fourth International Conference and Exhibition on Horizontal Well Technology, Calgary, Canada, Nov. 2000
- Stanford - Heriot-Watt Reservoir Description Forum, Carmel, CA, Aug. 2001 (two presentations)
- SIAM Conference on Mathematical and Computational Issues in the Geosciences (two presentations)
- First SIAM Conference on Computational Science and Engineering, Washington DC, Sept. 2000
- King Fahad University of Petroleum and Minerals, Saudi Arabia, March 2001

1 General Introduction and Overview

Reservoir simulation represents an essential tool for the management of oil and gas reservoirs. One of the most important issues in reservoir simulation is the representation of the well in the simulator and the linkage of the well to the reservoir. This issue is even more important in the modeling of non-conventional wells, which include horizontal, highly deviated, multilateral and other complex wells. Accurate tools for modeling such wells are needed, as the costs for drilling and completing non-conventional wells are very high. However, many of the current techniques for modeling non-conventional wells are inadequate, as these wells are significantly more complex than simple vertical wells.

Additional complexities, relative to vertical well modeling, arise in non-conventional well modeling for a number of reasons. Because non-conventional wells introduce a high degree of geometric complexity into the system, specialized gridding and discretization techniques are required to accurately model their performance. The effects of near-well heterogeneity can strongly impact non-conventional well performance, so fine scale permeability variation must also be represented in flow models. In addition, the effects of wellbore hydraulics (single phase or multiphase wellbore flows) can significantly affect the performance of long horizontal wells. This is in contrast to vertical wells, for which wellbore flow typically has little if any effect on productivity. Finally, stand-alone tools developed for modeling specific aspects of non-conventional well performance must be implemented into reservoir simulators. This requires that a research reservoir simulator capable of accommodating general well models be developed.

This project is directed toward significantly improving modeling approaches in three main areas. These include the development of (1) advanced reservoir simulation techniques for modeling non-conventional wells; (2) improved techniques for computing well productivity (for use in reservoir engineering calculations) and well index (for use in simulation models), including the effects of wellbore flow; and (3) accurate approaches to account for heterogeneity in the near-well region and throughout the reservoir. We now consider each of these general areas in more detail.

Non-conventional well modeling represents one component of the overall reservoir simulation. Because our research targets the development of an improved overall simulation capability, we are developing a general object-oriented research simulator. This simulator, described in Part I (Chapter 2) of this report, will accommodate general grids and a variety of procedures for treating the system variables. Specifically, the simulator allows for the use of different sets of primary and secondary variables as well as varying degrees of implicitness. For example, in a compositional simulation, pressure and saturation can be represented implicitly and all other variables implicitly. This is in contrast to standard approaches, which are either fully implicit or implicit in pressure only. We demonstrate improved efficiency relative to current approaches using our new formulation.

In Chapter 3, we present our recent work on the generation and use of flow-based grids for reservoir simulation. Flow-based grids can be generated either globally (for the entire reservoir) or modularly (for a single region or “module”) by solving a single phase flow problem. The streamlines and isopotentials from this flow problem provide a natural simulation grid with higher resolution in high flow regions and coarser resolution elsewhere. We demonstrate significantly improved results, over those attained using simple grids, with the flow-based grid generation procedure for both two and three dimensional flow problems.

As indicated above, wellbore hydraulics can impact the performance of non-conventional wells. A significant component of our research program is therefore directed toward the modeling of multiphase wellbore flows. To enable this research, we maintain a large database of experimental data collected over more than 40 years by a number of different research groups. Efficient and robust models for wellbore flow are required when these models are used within the context of reservoir simulation. This is because the wellbore flow model is coupled to the reservoir model and it is important that the wellbore model not act to significantly degrade the overall simulation efficiency. In Part II (Chapter 4) of this report, we present a procedure for determining the parameters of a drift-flux flow model from experimental data. The drift-flux modeling approach relates the in situ velocity of one of the phases to the mixture velocity. This model is currently used in reservoir simulators, and our new technique for representing model pa-

rameters is shown to improve the accuracy of the model for steady state flow calculations.

In Part III of this report we describe two procedures for efficiently representing the effects of reservoir heterogeneity in flow models. A general semi-analytical procedure for modeling the performance of non-conventional wells is presented in Chapter 5. This technique, based on Green's functions, is applicable only for single phase flow (i.e., primary production) problems. The semi-analytical method is highly efficient because it does not require that the reservoir be discretized. Rather, the well is discretized into some number of segments. The method accounts approximately for reservoir heterogeneity, the effects of wellbore hydraulics and the effects of downhole inflow control devices such as chokes. Results for several complex problems, involving, for example, a dual lateral well with downhole chokes in a fluvial reservoir, are presented.

In Chapter 6 we consider the upscaling of a fine scale permeability field onto a general finite difference simulation grid, such as the flow-based grids described in Chapter 3. We present a numerical technique, which applies a specialized finite element procedure, for computing the upscaled cell permeability tensor for general quadrilateral cells. The accuracy of the method is demonstrated through a number of example cases. Then, the method is applied in conjunction with a flow-based grid generation procedure. In this case, the overall approach displays a very significant improvement in accuracy relative to standard procedures. In future work we plan to extend this or a closely related procedure to the three dimensional case.

Part I. Development of Advanced Reservoir Simulation Techniques for Modeling Non-Conventional Wells

A primary component of our effort involves the development of a general purpose research simulator. This simulator will be used as a platform for research in a variety of areas, including general formulations, gridding and discretization. In Chapter 2 of this report, we describe our recent work addressing the formulation of the equations describing compositional flow. In Chapter 3, recent results on the development of flow-based grid generation procedures are reported.

2 Development of a General Purpose Research Simulator

by Hui Cao and Khalid Aziz

2.1 Introduction

Reservoir simulation is a critical tool for the optimum development of oil and gas fields. While reservoir simulators of various degrees of sophistication have been in use for over 40 years, they have become standard tools of the reservoir engineer over the last 20 years or so. With the expansion in the use of simulators, industry is demanding more and more versatile and faster simulators. Several companies are now exploring the development of the next generation of simulators to meet future needs.

Reservoir simulation research at a University like Stanford requires a versatile framework for developing and maintaining a modular General Purpose Research Simulator (GPRS). In a research simulator of this type modularity and ease of maintenance is of greater importance than speed. Eventually a simulator like this should allow researchers to explore different techniques for modeling oil recovery processes without undue need for programming from scratch. Such a GPRS is under development at Stanford University.

Since black oil models have already reached a high level of sophistication, initially the major focus of our research will be compositional modeling with Cartesian grids. Later we will include modules to handle:

- thermal processes,
- geomechanics,
- flexible grid,
- surface facilities,
- non-conventional wells,
- history matching, and
- optimization.

We also plan to do research on the development of models that can handle different oil recovery processes at the same time in different portions of the same field/reservoir. Our results should guide companies on the development of commercial software for the next generation of reservoir simulators. The primary goal of our work is the exploration of new, more efficient numerical techniques, not the development of code for direct commercial applications.

Compositional simulation with an arbitrary number of components is required for most EOR processes. It is more complex than standard black-oil simulation, due to the presence of a large number of components and various issues related to complex phase behavior (flash calculation, phase disappearance and reappearance) in such multicomponent, multiphase systems. Because of this complexity, a large number of compositional models have been proposed and many others are possible. Previous models are based on the use of different

- equations,
- variables, and
- level of implicitness used in solving the equations.

One of our objectives is to evaluate the performance of different (existing and new) compositional models. In order to do this and at the same time provide opportunities for research by different individuals on different aspects of reservoir simulation, a new general approach for formulating and solving reservoir simulation equations is developed. This approach makes it possible to develop and evaluate virtually any kind of model.

Since the pioneering work of Coats (1980) many papers have been published on various approaches to solving reservoir simulation equations for multicomponent systems. This research derives from the work of Coats (1980), Young and Stephenson (1983), Kendall et al. (1983), Nghiem et al. (1983), Acs et al. (1985), Chien et al. (1985), Watts (1986), Aziz and Wong (1988), Wong et al. (1990) and Coats (1998, 1999). Both the standard black-oil and the new compositional models have been implemented within the framework of GPRS. Initial testing is in progress.

Preliminary investigations have shown the possibility of a new compositional model that may be far more efficient than existing models. The new model solves pressure and saturations implicitly and we refer to it as the IMPS model. In conventional compositional simulations either just pressure is treated implicitly (IMPES or IMPEC) or all variables are treated implicitly (Fully Implicit or FI). In the first approach the timestep is highly restricted and in the second approach the amount of work per timestep increases sharply as the number of components needed to describe the system increases. Preliminary tests show that IMPS is more stable than the conventional IMPES model and the results are almost as good as those from the FI model. A new approach to handle phase disappearance and reappearance further enhances the efficiency of the proposed approach.

During this phase of our work the performance of several compositional models has been compared for a four component case. Future work will evaluate performance of different models for a broad range of cases. We will also explore techniques for automatically switching to the most efficient approach during the simulation process.

The next section will present the general formulation implemented in GPRS. In Section 2.3 we will present some preliminary results, followed by a discussion of future directions in Section 2.4.

2.2 General Formulation of GPRS

In this section we will give a brief description of the general formulation implemented in GPRS. In developing this general formulation the following points were considered:

- Reservoir simulation involves a large number of coupled nonlinear equations, and complexity increases dramatically as we go from black-oil to compositional models. For compositional models, direct solution of the equations resulting from the linearization process (Newton's method) for the FI model is impractical.
- Generally, there is no need to solve all of the coupled equations together. For each simulation model, there are a minimum number of independent variables and equations that determine the thermodynamic state (intensive and extensive) of the system; these variables are referred to as the primary variables. For isothermal compositional systems, the extensive thermodynamic state is determined by n_c variables, i.e., the number of components (balance equations) in the system. By solving for this minimum number of variables first, we can save a large amount of computation time. Even so, fully implicit compositional simulation is still too expensive in most cases.
- We can further reduce the cost by treating some variables explicitly. But this simplification will limit the maximum time step size, and may cause balance errors (mass or volume) in the simulation. There are numerical techniques to reduce or switch the balance errors (Coats et al., 1998)

Figure 2.1: Reduction process

Based on the above considerations, we follow the approach shown in Fig. 2.1. We start with the full set of nonlinear equations (general formulation), then reduce them to the primary set (minimum number), and finally reduce the implicit level to further speed

up the simulation process. This reduction procedure is carried out in the new General Formulation Approach.

Another use of this General Formulation Approach is that it can be used to derive any kind of models through a unified process. The requirement of this approach is that all models share the same set of primary equations (here we select the n_c mass balance equations), which is the case for most models, including the volume balance type of models. We first define a base model, which uses variables that facilitate the calculation of derivatives (natural variables). Then other models are built from this base model through a unified solution procedure.

We will first present the general nonlinear equation set that is fundamental to this approach. Then the base model will be presented and the process required for variable switching will be illustrated. Finally, the unified solution procedure will be presented. It is a two step reduction, first the total set is reduced to the primary set, then the implicit level is reduced to further speed up the computation.

The General Nonlinear Equation Set

The most general nonlinear equation set can be expressed as,

$$\vec{F}(\vec{X}) = 0 \quad (2.1)$$

where $\vec{F}$ is the full set of equations and $\vec{X}$ is the full set of variables. Next we divide the equations and variables into primary and secondary parts, and rewrite Eq. 2.1 as:

$$\begin{cases} \vec{F}_p(\vec{X}_p, \vec{X}_s) = 0 \\ \vec{F}_s(\vec{X}_p, \vec{X}_s) = 0 \end{cases} \quad (2.2)$$

where subscript p stands for primary, and s stands for secondary.

For isothermal compositional models where hydrocarbons and water are totally separated, $\vec{F}_p$ will be the mass balance equations for each hydrocarbon component and for water:

$$F_{c,i} = \frac{\partial}{\partial t} [V\phi(S_o\rho_o x_c + S_g\rho_g y_c)]_i - \sum_l [T(\lambda_o\rho_o x_c \Delta\Phi_o + \lambda_g\rho_g y_c \Delta\Phi_g)]_{i,l} + \sum_w (\rho_o x_c q_o^w + \rho_g y_c q_g^w)_i = 0$$

$$c = 1, \dots, n_h \quad (2.3)$$

$$F_{w,i} = \frac{\partial}{\partial t} [V\phi S_w \rho_w]_i - \sum_l (T\lambda_w \rho_w \Delta\Phi_w)_{i,l} + \sum_w (\rho_w q_w^w)_i = 0 \quad (2.4)$$

Optionally, we can replace one of the hydrocarbon component mass balance equations by a total hydrocarbon mass balance equation.

$$F_{h,i} = \frac{\partial}{\partial t} [V\phi(S_o\rho_o + S_g\rho_g)]_i - \sum_l [T(\lambda_o\Delta\Phi_o + \lambda_g\Delta\Phi_g)]_{i,l} + \sum_w (\rho_o q_o^w + \rho_g q_g^w)_i = 0 \quad (2.5)$$

$\vec{F}_s$ are the phase equilibrium relations:

$$F_e = f_{c,o} - f_{c,g} = 0, \quad c = 1, \dots, n_h \quad (2.6)$$

The same nonlinear equation set will be used for all types of models. For the base model, the full set of variables, $\vec{X}_{base}$, is chosen to be:

$$p \equiv p_g,$$

$$S_g, S_o,$$

$$y_c, x_c, c = 1, 2, \dots, n_h - 1,$$

These are actually the natural variables for this problem (Coats, 1980).

Here we have a total of $2n_h + 1$ variables in $2n_h + 1$ nonlinear equations. Capillary pressure constraints are used to remove water and oil pressure, the saturation constraint is used to remove water saturation, and mole fraction constraints are used to remove the phase mole fractions of the last hydrocarbon component.

The black oil model is treated as a simplified case of the general compositional model, where flash calculation is no longer needed and mole fractions are either constant

or only functions of pressure. For thermal compositional models, $\bar{F}_p$ will include the energy balance equation, and $\bar{X}_p$ will include temperature or internal energy.

Switching To New Variables $\bar{X}$

We have our base model, and we know how to calculate the derivatives with respect to its variables, which are $\frac{\partial \bar{F}}{\partial \bar{X}_{base}}$. Now we need to calculate the derivatives with respect to

new variables, which are $\frac{\partial \bar{F}}{\partial \bar{X}}$. We can do this by the chain rule $\frac{\partial \bar{F}}{\partial \bar{X}} = \frac{\partial \bar{F}}{\partial \bar{X}_{base}} \frac{\partial \bar{X}_{base}}{\partial \bar{X}}$,

provided we know $\frac{\partial \bar{X}_{base}}{\partial \bar{X}}$, which are determined from the relations between $\bar{X}$ and $\bar{X}_{base}$. The entire switching process has the following steps:

- 1) Build the explicit relation between $\bar{X}$ and $\bar{X}_{base}$, and calculate the transformation

$$\text{matrix } TRAN = \frac{\partial \bar{X}_{base}}{\partial \bar{X}}$$

- 2) Calculate the new derivatives by $\frac{\partial \bar{F}}{\partial \bar{X}} = \frac{\partial \bar{F}}{\partial \bar{X}_{base}} \cdot TRAN$

- 3) Solve the linear system, get $\delta \bar{X}$ and update $\bar{X}^{v+1} = \bar{X}^v + \delta \bar{X}$

- 4) Update base model variables $\bar{X}_{base}^{v+1} = \bar{X}_{base}^v + \delta \bar{X}_{base}^v = \bar{X}_{base}^v + TRAN \cdot \delta \bar{X}$

Each model has its own transformation matrix (due to different relations between $\bar{X}$ and $\bar{X}_{base}$), but all of the steps are exactly the same. Based on a review of existing compositional models, possible choices for $\bar{X}$ (other than base model variables) are:

$$S_w, x_{n_h}, y_{n_h}, z_c, W, F, l$$

They can be related to the base model variables by:

$$S_w = 1 - S_g - S_o \quad (2.7a)$$

$$x_{n_h} = 1 - \sum_{c=1}^{n_h-1} x_c \quad (2.7b)$$

$$y_{n_h} = 1 - \sum_{c=1}^{n_h-1} y_c \quad (2.7c)$$

$$z_c = \frac{\rho_o S_o x_c + \rho_g S_g y_c}{\rho_o S_o + \rho_g S_g} \quad (2.7d)$$

$$W = \rho_w (1 - S_g - S_o) \quad (2.7e)$$

$$F = \rho_o S_o + \rho_g S_g \quad (2.7f)$$

$$l = \frac{\rho_o S_o}{\rho_o S_o + \rho_g S_g} \quad (2.7g)$$

where $\rho_w = \rho_w(p)$, $\rho_o = \rho_o(p, x_1, \dots, x_{n_h-1})$, $\rho_g = \rho_g(p, y_1, \dots, y_{n_h-1})$. Note that this process is equivalent to the use of chain rule to get the derivatives.

Now we have the full set of equations and variables, and we also have different choices of variables. The next step is to reduce the number of variables and equations to be solved. First we reduce it to the minimum number (n_c) of equations and variables required to establish the thermodynamic state (fully implicit model), then we can further reduce the implicit level.

Reduce Full Set $\vec{F}(\vec{X}) = 0$ to Primary Set $\vec{F}_p(\vec{X}_p) = 0$

In order to write an iterative process to solve $\vec{F}(\vec{X}) = 0$, we can use the Newton-Raphson method, or we can express $\vec{F}(\vec{X})$ in Taylor series.

$$0 = \vec{F}^{n+1}(\vec{X}) \approx \vec{F}^{v+1}(\vec{X}) = \vec{F}^v(\vec{X}) + \frac{\partial \vec{F}^v}{\partial \vec{X}} \delta \vec{X} \quad (2.8)$$

It can be easily demonstrated that these two methods are equivalent.

In the following development, Taylor series are used to derive the primary set. We will show two ways (analytical derivation and matrix manipulation) to achieve the same final goal.

Analytical Derivation

Let us define the following matrices:

$$A = \frac{\partial \vec{F}_p}{\partial \vec{X}_p}, B = \frac{\partial \vec{F}_p}{\partial \vec{X}_s}, C = \frac{\partial \vec{F}_s}{\partial \vec{X}_p}, D = \frac{\partial \vec{F}_s}{\partial \vec{X}_s} \quad (2.9)$$

and vectors:

$$\vec{M} = \vec{F}_p, \vec{N} = \vec{F}_s, \quad (2.10)$$

where A, B, C, D together form the full Jacobian matrix for Eq. 2.2, and $\vec{M}, \vec{N}$ are the corresponding right hand sides (RHS), as shown on the first frame of Fig. 2.2.

The first step is to relate secondary variables to primary variables. We start with the secondary equation set: $\vec{F}_s(\vec{X}_p, \vec{X}_s) = 0$. Use of Taylor series on this equation set yields,

$$0 \approx \vec{F}_s^{v+1}(\vec{X}_p, \vec{X}_s) = \vec{F}_s^v(\vec{X}_p, \vec{X}_s) + \frac{\partial \vec{F}_s^v}{\partial \vec{X}_p} \delta \vec{X}_p + \frac{\partial \vec{F}_s^v}{\partial \vec{X}_s} \delta \vec{X}_s = \vec{N}^v + C^v \delta \vec{X}_p + D^v \delta \vec{X}_s \quad (2.11)$$

Rearranging Eq. 2.11, we get:

$$\delta \vec{X}_s = -(D^{-1}C)^v \delta \vec{X}_p - (D^{-1}\vec{N})^v \quad (2.12)$$

The second step is to write primary equations in primary variables. Now we use Taylor series on the primary equation set: $\vec{F}_p(\vec{X}_p, \vec{X}_s) = 0$ to get

$$0 \approx \vec{F}_p^{v+1}(\vec{X}_p, \vec{X}_s) = \vec{F}_p^v(\vec{X}_p, \vec{X}_s) + \frac{\partial \vec{F}_p^v}{\partial \vec{X}_p} \delta \vec{X}_p + \frac{\partial \vec{F}_p^v}{\partial \vec{X}_s} \delta \vec{X}_s = \vec{M}^v + A^v \delta \vec{X}_p + B^v \delta \vec{X}_s \quad (2.13)$$

By substituting Eq. 2.12 into the above equation, we get,

$$0 = \vec{M}^v + A^v \delta \vec{X}_p + B^v [-(D^{-1}C)^v \delta \vec{X}_p - (D^{-1}\vec{N})^v] \quad (2.14)$$

Rearranging terms yields,

$$(A - BD^{-1}C)^v \delta \vec{X}_p = -(\vec{M} - BD^{-1}\vec{N})^v \quad (2.15)$$

This is our final fully implicit system, which only includes primary variables.

Matrix Manipulation (Gaussian Elimination)

We can get the same Eq. 2.15 by using Gaussian elimination on blocks of the full Jacobian matrix. This is illustrated in Fig. 2.2.

Figure 2.2: Reduce to primary set by Gaussian elimination

In the first step, the secondary equation set is multiplied by D^{-1} . In the second step, the secondary equation set is multiplied by B . Finally the secondary equation set is subtracted from the primary equation set. We can now write out the final primary equation set as:

$$(A - B D^{-1} C) \delta \bar{X}_p = -(\bar{M} - B D^{-1} \bar{N})$$

which is the same as Eq. 2.15.

Note that, matrices A and B are created by the flow equations, and they exist on both diagonal and off-diagonal parts (flux terms) of the full Jacobian matrix. While matrices C and D are created by phase equilibrium relations, and they only exist on the diagonal part. Matrix D is actually the Jacobian matrix for flash calculations. When we update the secondary variable, it is equivalent to doing one flash calculation per Newton iteration.

We need to eliminate the matrix B on both the diagonal part and the off-diagonal part. For diagonal B , we use matrices C and D of its own block. For off-diagonal B , we need to use C and D which are on the same column as B . Basically, the rule is that matrix A , B , C and D are always the derivatives respect to the variables in the same gridblock:

Figure 2.3: Process of removing diagonal and off-diagonal Bs

For a system in Fig. 2.3, the reduction calculations are (reduce Bs to 0):

$$\begin{aligned} A1 &= A1 - B1 \cdot D1^{-1} \cdot C1, & M1 &= M1 - B1 \cdot D1^{-1} \cdot N1 \\ A12 &= A12 - B12 \cdot D2^{-1} \cdot C2, & M1 &= M1 - B12 \cdot D2^{-1} \cdot N2 \end{aligned}$$

$$\begin{aligned} A2 &= A2 - B2 \cdot D2^{-1} \cdot C2, & M2 &= M2 - B2 \cdot D2^{-1} \cdot N2 \\ A21 &= A21 - B21 \cdot D1^{-1} \cdot C1, & M2 &= M2 - B21 \cdot D1^{-1} \cdot N1 \end{aligned}$$

The As and Ms are extracted out for fully implicit solve.

If we treat transmissibility explicitly, then there is no need for the reduction of off-diagonal Bs, because flux terms do not have derivatives with respect to secondary variables (pressure is always a primary variable), and all off-diagonal Bs are zero. After we solve for the primary variables, the secondary variables are calculated explicitly. During the calculations, D^{-1} is calculated and stored in D , and BD^{-1} is calculated and stored in B . The updated As and Ms are all stored in their old locations.

Up until now, we have derived the fully implicit Jacobian matrix. For fully implicit simulation we can just stop here. But in lots of situations, we desire to reduce the implicit level to speed up the simulation process. The next section will show how this is done.

Reduce the Implicit Level of Fully Implicit System $\vec{F}_p(\vec{X}_p) = 0$

For each grid block, the fully implicit Jacobian matrix is of the following form,

	diagonal $V_1 V_2 V_3 P$		One neighbor $V_1 V_2 V_3 P$	
	X X X X X X X X X X X X X X X X		X X X X X X X X X X X X X X X X	

Figure 2.4: Jacobian matrix of one block (fully implicit)

where P represents pressure and other implicit variables, V_1 , V_2 , and V_3 are the explicit primary variables.

We can reduce the implicit level of the Jacobian matrix in two steps:

- 1) First set all flux term derivatives with respect to explicit primary variables to zero, or simply don't calculate them. This is illustrated in Fig. 2.5, where pressure is the only implicit variable.

	diagonal						one neighbor				
	V ₁	V ₂	V ₃	P			V ₁	V ₂	V ₃	P	
	X	X	X	X			0	0	0	X	
	X	X	X	X			0	0	0	X	
	X	X	X	X			0	0	0	X	
	X	X	X	X			0	0	0	X	

Figure 2.5: Jacobian matrix of one block with only pressure implicit

- 2) Next, we use the IMPES reduction factor (Coats, 1999) or Gaussian elimination to remove the diagonal part of derivatives with respect to explicit primary variables from one of the primary equations (the last equation in Fig. 2.5).

diagonal V ₁ V ₂ V ₃ P				one neighbor V ₁ V ₂ V ₃ P			
X	X	X	X	0	0	0	X
0	X	X	X	0	0	0	X
0	0	X	X	0	0	0	X
0	0	0	X	0	0	0	X
.....							

Figure 2.6: Final reduced Jacobian matrix of one block

Fig. 2.6 shows the process of using Gaussian elimination to get the upper triangular matrix. The last equation could now be extracted out to solve for pressures implicitly.

The procedure of using the IMPES reduction factor is shown on Fig. 2.7. It is similar to the procedure in Fig. 2.2. First we divide the diagonal and off-diagonal blocks as shown in Fig. 2.7. Each block is divided into four parts. Then we move all of the non-zero parts to individual matrices, as shown in Fig. 2.7. Next we use the procedure of Fig. 2.2 to reduce the lower left part (matrix C) of the diagonal block to zero. The IMPES reduction factor is the corresponding $-CA^{-1}$ matrix (for this case, it is a vector, because C is a vector and A is a matrix). IMPES reduction factor could be viewed as Gaussian elimination performed on a block matrix. For the IMPES model (only pressure implicit), the IMPES reduction factor is a vector, basically it assigns a scalar to each equation, then sums all equations to remove all explicit variables. Coats (1999) has also analyzed the physical meaning of the IMPES reduction factor.

Figure 2.7: IMPES Reduction Factor

Here we need to operate on the diagonal part, off-diagonal part and RHS at the same time. The rule is that we operate on the equations belonging to the same gridblock at one time. This is illustrated in more detail in Fig. 2.8:

Figure 2.8: Illustration of IMPES Reduction

We do the following calculations (remove C):

$$D = D - C \cdot A^{-1} \cdot B$$

$$F = F - C \cdot A^{-1} \cdot E$$

$$N = N - C \cdot A^{-1} \cdot M$$

then D , F and N are extracted out to form the smaller implicit system. After we get the implicit variables, the explicit variables are updated explicitly by:

$$\delta \tilde{X}_{\text{exp}} = A^{-1} (M - B \cdot \delta \tilde{X}_{\text{imp}} - E \cdot \delta \tilde{X}_{\text{imp}}')$$

This General Formulation Approach allows us to get a model with any selection of variables and implicit level. All of the operations are performed on a gridblock by gridblock basis, so it would be a natural extension of this process to use different implicit levels for different gridblocks (as in the Adaptive Implicit Method or AIM). This approach also allows the use of different models for different reservoirs (reservoirs are only connected through well groups and boundary interfaces in parallel computation).

The General Formulation Approach only defines the general steps and methods to reach the final goal, and they are suitable for any model. But in order to achieve maximum efficiency, when implementing this approach, we need to consider details for each specific model.

2.3 Preliminary Results

Two cases have been studied, a black-oil simulation using a compositional formulation, and a series of compositional simulations using different models and different implicit levels. Both cases used the same 5 by 5 by 5 homogeneous reservoirs, and their major difference was the fluid model.

The black-oil simulation was used to validate this simulator, and its results were compared with Eclipse100 (GeoQuest, 2000) results. The compositional simulations were used to test the General Formulation Approach and evaluate the performance of different models. First, the results using fully implicit Coats (1980) type variables were compared with Eclipse300 (GeoQuest, 2000) results to validate the compositional model. Then the results for different variable selection and implicit levels were compared with the base results where we solved the full set of equations and variables together (an expensive but accurate process).

For both simulations, the convergence of Newton iterations is tested by both the maximum residue and the maximum change in variables of each gridblock. We require that:

- All normalized residues be less than 0.1. Flow equation residues are normalized by the accumulation terms, and phase equilibrium residues are normalized by component fugacities.
- The maximum absolute pressure change divided by the average reservoir pressure be less than 0.0001.
- The maximum absolute saturation change is set at 0.005.
- The maximum absolute mole fraction change is set at 0.001.

Figure 2.9: Reservoir illustration

Black Oil Results

This was a water/oil black-oil simulation of a quarter of a 5-spot. One water injector was located at (1, 1, 5), and one oil producer was located at (5, 5, 1), and both wells were at constant rate control. Initially, the reservoir contained only oil (Fig. 2.9a). The simulation was run to 3000 days with a maximum timestep size of 30 days. GPRS took an average of 1.1 Newton iterations per timestep, and Eclipse100 (GeoQuest, 2000) took an average of 1.3 Newton iterations per timestep. Eclipse100 uses only the maximum saturation normalized residual for convergence control (Eclipse Technical Description, 2000), and it sets this limit to 0.001, which is stricter than that used in GPRS. Hence, Eclipse needs a few more Newton iterations. Fig. 2.10 shows our results, compared with Eclipse100.

Figure 2.10: Block pressure and saturation at injector and producer (black-oil)

GPRS results are drawn in dots and Eclipse100 results are drawn in solid lines. Fig. 2.10a compares the block pressure at both wells. Fig. 2.10b compares the well block water saturations. The black lines (lower) are for the producer and the blue lines (upper) are for the injector. Basically, these two results are on top of each other, i.e., agreement between GPRS and Eclipse100 is excellent.

This confirms that the basic modules of GPRS are working correctly. The next step is to further validate our simulator with compositional problems. After that, we can start the work of evaluating different compositional models.

Compositional Model Results

This was a four-component, two-phase (Gas-Oil) compositional simulation. One producer was located at (5, 5, 1), which was under bottom hole pressure control (Fig. 2.9b). The Peng-Robinson (1976) Equation of State was used for flash calculations. Initially the reservoir had both oil and gas phase. All of the simulations were run to 1000 days with a maximum timestep size of 30 days. We first compare GPRS results with Eclipse300 results, then evaluate the performance of different models.

Validation of compositional modeling

Let's compare GPRS results with Eclipse300 (GeoQuest, 2000) results. GPRS uses the Coats (1980) type variables, while Eclipse300 uses pressure and overall molar density of

each component (Eclipse Technical Description, 2000). Both simulators run in fully implicit mode. GPRS took an average of 1.5 Newton iterations per timestep, while Eclipse300 used an average of 1.1 Newton iterations per timestep. The slightly higher number of Newton iterations in GPRS was caused by the differences in convergence control. In Eclipse300, only the maximum changes of pressure and effective saturations (the effective saturation change for a given molar density change) are used for convergence control (Eclipse Technical Description). Also these limits are higher than the limits used in GPRS (Eclipse300 uses 0.1 atm for absolute pressure change, and 0.01 for effective saturation change). Fig. 2.11 compares GPRS results with Eclipse300 results.

Figure 2.11: Block pressure and saturation comparison (compositional)

Similar to Fig. 2.10, GPRS results are drawn in dots and Eclipse300 results are drawn in solid lines. Fig. 2.11a compares the pressure at block (1,1,5) and producer (5,5,1). Fig. 2.11b compares the oil saturations of these two blocks. The black lines (lower lines) are for the producer (5,5,1) and the blue lines (upper lines) are for block (1,1,5). The match of oil saturations at the producer block is not exact. The reason for this is that Eclipse300's convergence criteria (pressure and effective saturation changes) are not strict enough for this case. Actually Eclipse goes to the next timestep without full convergence in the current timestep, which cause some mass balance errors. If we reduce the maximum timestep size from 30 days to 10 days in Eclipse300, it should help it to converge better. The results are shown in Fig. 2.12. Here Eclipse300 uses a maximum timestep size of 10 days, and GPRS still uses 30 days.

Figure 2.12: Block pressure and saturation comparison (Eclipse300 uses 10 days)

The match is now excellent. Based on these tests we can say that the compositional modeling in GPRS is working fine, and in some cases it even gives better results than Eclipse300 due to a stricter convergence control. Now that the compositional model and the General Formulation Approach have been tested, we can start the evaluation of the performance of difference compositional models.

Evaluation of different compositional models

The full set of equations and variables was directly solved (an expensive calculation in general; here it took 340 seconds to complete), and its results were used as the base results to compare with the results of other cheaper approaches.

For this particular problem, we have 4 hydrocarbon components and 2 hydrocarbon phases. So the full set of equations (total of 8) will include the mass balance equation and the equal fugacity relations for each hydrocarbon component. Among them, the four mass balance equations are the primary equations.

For variable type 1 models, the full set of variables are p_g , S_g , y_1 , y_2 , y_3 , x_1 , x_2 , x_3 , where y_i are the component mole fractions in the vapor (gas) phase and x_i are the component mole fractions in the liquid (oil) phase, and p_g , S_g , y_1 and y_2 are the primary variables. Three different implicit levels are used for the primary variables. In the first case,

p_g , S_g , y_1 and y_2 are implicit (fully implicit), only pressure is implicit (IMPES) for the second case, and pressure and saturation are implicit (IMPS) for the last case.

For variable type 2 models, the full set of variables are p_g , z_1 , z_2 , z_3 , y_3 , x_1 , x_2 , x_3 , where z_i are the overall component mole fractions, and p_g , z_1 , z_2 and z_3 are the primary variables. Two different implicit levels are used for the primary variables. For the first case, p_g , z_1 , z_2 and z_3 are implicit (fully implicit), while only pressure is implicit (IMPES) for the second case (sometimes referred to as IMPEM or IMPEC). Next, we will compare the results of each model with the results of the base model.

- ***Fully implicit (variable type 1)***

Starting from variable type 1 (base variables in GPRS), we can easily get the primary set from the full set by matrix manipulation and use it as the fully implicit model. This fully implicit model is the most stable one, but it takes the longest time to solve (four variables per gridblock solved simultaneously). The results of gas and oil phase volumetric flow rate at the producer block are shown in Fig. 2.13. The base case results are drawn in solid lines, and the fully implicit results are drawn in dots. The blue line and the blue dots (upper one) are for gas rate, and the black line and the black dots (lower one) are for the oil rate. Gas rate (SCF/Day) uses the left y-scale, and oil rate (SCF/Day) uses the right y-scale. All of the other results in this part will be shown in the same way.

Here, a maximum timestep size of 30 days was used, and it took an average 1.5 Newton iterations per timestep for a total of 39 timesteps, and the total running time was around 16.9 seconds. Basically, the fully implicit results are the same as for the base case, because they the two represent the same system but solved differently.

Figure 2.13: Fully implicit results for variable type 1

- **IMPES (variable type 1)**

After we have the fully implicit model, we can further reduce the implicit level of certain primary variables. Here we only keep pressure as the implicit variable, and the other three variables are treated explicitly. This explicit treatment will reduce the number of unknowns we need to solve simultaneously, hence the computation time per Newton iteration will be much smaller. On the other hand, treating some variables explicitly causes a limitation on the maximum allowable timestep size. In some cases Newton iterations are not performed in IMPES, which causes some mass balance errors. Let's see the IMPES results with no iteration. This run used 56 timesteps with a maximum timestep size of 30 days, and it took 3.7 seconds to finish. The results are shown in Fig. 2.14.

Figure 2.14: IMPES results for variable type 1 (max 30 days)

We can see that they roughly agree with the base results and may be acceptable for some engineering purposes, but they clearly show significant errors. There are a lot of fluctuations for later timesteps, and the results have serious problems around 350 days, when the oil flow rate and saturation start to increase. All of these problems are caused by the mass balance error which is passed from timestep to timestep. Normally we can control this error by using smaller timestep size. Of course we can also use the volume balance concept to correct this mass balance error. First we limit the maximum timestep size to 10 days and do the same simulation. The results for this case are shown in Fig. 2.15. We can see that while the results have improved significantly, there is still some problem at around 350 days. Due to the smaller timestep size, the computational cost increased, and this run was finished in 6.9 seconds with 105 timesteps.

Figure 2.15: IMPES results for variable type 1 (max 10 days)

- *IMPS (variable type 1) --- new model*

Now, let's try the IMPS model, the implicit saturation treatment should help us remove the problem around 350 days. Here, we first run this model without Newton iteration, then with Newton iteration (full convergence). Both runs used a maximum timestep size of 30 days.

The run without Newton iteration took 3.7 seconds with 47 timesteps, and the results are shown in Fig. 2.16. We can see that treating saturation implicitly makes the problem much more stable, the results are excellent and the problem around 350 days has totally disappeared. The improved stability of IMPS made it possible to do the same simulation in fewer timesteps (47) than by the IMPES method (56). The reduction in the number of timesteps was enough to offset the extra cost of treating saturations implicitly. The total running time was about the same as for the IMPES method, but there is a huge improvement in results compared with those from the IMPES model.

Figure 2.16: IMPS results (no iteration)

From Fig. 2.16, we notice that the match is not perfect, the mass balance errors are caused because in this case Newton iterations are not performed. Next we run the IMPS model with iteration until full convergence, just as was the case for the fully implicit model. This run took an average of 1.5 Newton iterations with 46 timesteps, and the whole run was finished within 8.6 seconds. The results are shown in Fig. 2.17. As expected, the match is perfect. Actually, for any converged model, the result should match the base case results. Regardless of the method of solution, at convergence when all of the nonlinear equations are satisfied (zero residue), we should get the same result. The only question here is: does the method converge and how fast is the convergence? Since, for this problem, the IMPS Jacobian is near enough to the fully implicit Jacobian, the convergence is fast (same number of Newton iteration per timestep as for the fully implicit model). Of course, further tests with larger timesteps are needed. We also notice that, due to the explicit treatment of mole fractions, IMPS used more timesteps (46) than the fully implicit model (39).

Figure 2.17: IMPS results (iterate until converge)

Next, we investigate the use of variable type 2 models. The full set of variables is p_g , z_1 , z_2 , z_3 , y_3 , x_1 , x_2 , x_3 . Here we need to switch variables. After variable switching, all of the other steps are the same as for variables type 1 models.

- **Fully implicit (variable type 2)**

We use the fully implicit model with p_g , z_1 , z_2 and z_3 as primary variables. This simulation took the same number of Newton iterations and timesteps as the fully implicit model of variable type 1. The total running time was 17.1 seconds, a slight increase due to the variable switching process. The results are shown in Fig. 2.18. Basically, it is a perfect match, because we are solving the same system.

Figure 2.18: Fully implicit results for variable type 2

- **IMPES (variable type 2)**

Now let's try the IMPES model, where first p_g is solved implicitly, then z_1 , z_2 and z_3 are updated explicitly. As before, no Newton iteration is performed. This run took 3.8 seconds with 55 timesteps. The results are shown in Fig. 2.19. The match is not very good, but better than the IMPES model of variable type 1. The differences lie in the order in which the variables were updated (we get the same pressure solution, then update other variables in different orders). Further investigations are needed to fully understand this behavior.

Figure 2.19: IMPES results for variable type 2

2.4 Preliminary Conclusions and Future Directions

Preliminary Conclusions

The design of GPRS and basic implementations have been finished and tested. Other researchers can now start working on individual modules. Both black-oil and compositional models have been tested and validated. The general formulation approach implemented in GPRS makes it possible to develop an optimum strategy for the selection of variables and their implicit treatment. A new model using implicit pressure and saturations (IMPS) is proposed and evaluated. Furthermore, a new approach for handling phase disappearance and reappearance has been developed to avoid the inconvenience of changing equation in variable type 1 models.

Different compositional models, varying in their selection of variables and implicit levels, are evaluated using a four component system. Based on these preliminary tests, we can draw the following conclusions:

- **Fully Implicit models.** For both types of variable selection, fully implicit models are the most accurate and stable (making large timesteps possible). But the cost of simulation with these models is the highest, and the costs increase exponentially with the number of component. There are no significant differences in results or execution time between the fully implicit models with different variable selections.
- **IMPES models.** For both types of variables investigated, the IMPES models were tested with and without Newton iterations. They are fast, because they solve only one of the unknowns implicitly, regardless of the number of components in the system. But without Newton iterations they are not accurate, due to the mass balance error as a result of lack of convergence on the accumulation terms. In order to limit this error, we need to either correct for it in some way or use smaller timesteps. IMPES results for different variable selections have moderate differences, and the reason lies in the way that explicit variables are updated. Further investigations are needed to fully understand this.
- **IMPS model.** This new model can be run with or without Newton iteration. IMPS without Newton iteration is as fast as IMPES. The increased stability and larger timestep size are strong enough to offset the extra time spent on linear solves. At the same time, the results are much improved over IMPES results, due to the implicit treatment of saturations. For IMPS with Newton iteration, the results are the same as for the fully implicit models, and this model is much faster than fully implicit models. This advantage will be even greater for a large number of components. The total number of Newton iteration is roughly the same as for the fully implicit model, because the Jacobian matrix of IMPS is a good enough approximation to the fully implicit Jacobian. At the same time, the cost of each Newton iteration (time for linear solves) is much smaller than the fully implicit models.

In summary we can say that, based on our tests, the newly developed IMPS model without iteration could be viewed as an improved IMPES model, and IMPS with iteration could be viewed as an improved (faster) fully implicit model.

For the Adaptive Implicit Method (AIM), we may be able to use IMPS without iteration for the explicit gridblocks, and IMPS with iteration for the implicit gridblocks. This IMPS based AIM should have better performance than the old AIM. In this IMPS based AIM, all gridblocks will use the same IMPS formulation, and the only difference will be the number of iterations. , Hence, this AIM could be viewed as a natural extension of the IMPS model.

The procedure for implementing this new AIM will be to perform the following operations for each timestep:

- 1) Generate IMPS system for all gridblocks.
- 2) Solve the linear system and update variables (equivalent to IMPS without iteration).
- 3) Remove the gridblocks that are to be treated explicitly (based on some criteria), and only generate the IMPS system for the remaining gridblocks. The general formulation in GPRS can easily do this, because all operations are performed on a gridblock by gridblock basis.
- 4) For the remaining gridblocks solve the linear system and update the Jacobian and iterate to convergence.

We can remove a gridblock, if

- the variables in that gridblock have converged, and
- if all of its neighbors are have already been removed.

The IMPS system for removed gridblocks will not change. By doing so, the whole IMPS system will continue to reduce in size and we will be solving a smaller and smaller system of equations at each iteration. If the size reduction is small, then it will not be worthwhile to do this extra work.

Future Directions

- Evaluate the performance of different models under different conditions using harder and more complicated test problems. Further evaluate the advantages of the new IMPS model. At the same time, implement a mass balance correction procedure to control the mass balance error for the cases when no Newton iterations are performed.

- Implement and test the new IMPS based AIM and compare its results and speed with the traditional AIM.
- Develop an optimum strategy for automatically switching models.

2.5 References

- Acs, G., Doleschall, S., and Farkas, E.: "General Purpose Compositional Model," Soc. Pet. Eng. J. 25 (August 1985) 543-553.
- Aziz, K. and Wong, T.W.: "Considerations in the Development of Multipurpose Reservoir Simulation Modes," Proceeding of First and Second International Forum on Reservoir Simulation: Alpbach, Austria, Sept. 12-16, 1988 and Sept. 4-8, 1989.
- Chien, M. C. H., Lee, S. T. and Chen, W. H.: "A New Fully Implicit Compositional Simulator," presented at Eighth SPE Symposium on Reservoir Simulation, Dallas, Feb. 1985.
- Coats, K. H.: "An Equation of state Compositional Model," Soc. Pet. Eng. J. (October 1980) 363-376.
- Coats, K. H., Thomas, L. K. and Pierson, R. G.: "Compositional and Black-Oil Reservoir Simulation," SPE50990, 1998.
- Coats, K. H.: "A Note on Impes and Some Impes Based Simulation Models," SPE 49774, presented at SPE Reservoir Simulation Symposium, Houston, TX, 14-17 February 1999.
- GeoQuest, Schlumberger, Eclipse 100 and Eclipse 300 User manuals, 2000.
- Kendall, R. P., Morrell, G. O., Peaceman, D. W., Silliman, W. J., and Watts, J. W.: "Development of a Multiple Application Reservoir Simulator for Use on a Vector Computer," presented at SPE Middle East Oil Technical Conference, Manama, Bahrain, March 1983, No. 11483.
- Nghiem, L. X., Aziz, K., and Li, Y. K.: "A Robust Iterative Method for Flash Calculations Using the Soave-Redlich-Kwong or the Peng-Robinson Equation of State," Soc. Pet. Eng. J. (June 1983) 521-530.
- Peng, D. Y. and Robinson, D. B.: "A New Two-Constant Equation of State," Ind. Eng. Chem. Fundam. (1976), 15, 59-64.
- Watts, J. W.: "A Compositional Formulation of the Pressure and Saturation Equations," SPERE 1 (May 1986) 243-252.
- Wong, T. W., Firoozabadi, A., and Aziz, K.: "Relationship of the Volume-Balance Method of Compositional Simulation to the Newton-Raphson Method," SPE 18424, 1990.
- Young, L. C. and Stephenson, R. E.: "A Generalized Compositional Approach for Reservoir Simulation," Soc. Pet. Eng. J. 23 (October 1983) 727-742; Trans., AIME 275.

3 Flow-Based Modules for Grid Generation in Two and Three Dimensions

by Alexandre Castellini, Michael G. Edwards and Louis J. Durlofsky

3.1 Introduction

Geostatistical reservoir descriptions typically contain two to three orders of magnitude more grid blocks than can be accommodated by flow simulators. Therefore, prior to flow simulation, a coarsened reservoir model must be generated. This requires that a coarse scale grid be formed and that coarse scale grid block properties (such as permeability) be computed. One of the primary aims of a modern grid generator is to concentrate grid nodes in crucial regions of the flow domain. Ideally, for evolutionary problems, the grid should be dynamic and able to readjust as steep flow gradients propagate across the domain (e.g., Edwards, 1996). However, if the grid is static (fixed in time), as is the norm for the majority of current simulators, then the best strategy is less obvious. While one possibility is to adjust the grid density according to fine scale permeability variation, this will not account for permeability connectivity in the reservoir.

A basic observation is that the grid should have a concentration of nodes around geometric and geological features that strongly influence fluid flow. This suggests that the grid should have knowledge of flow response and permeability connectivity. Thus, while gridding of critical reservoir features such as wells, faults and bedding boundaries is clearly important, the local grid density should also be a function of the accessibility of a given reservoir region to fluid flow.

A number of methods for generating coarse scale grids with nodes distributed according to single phase flow response information have been proposed (e.g., Tran, 1995; Durlofsky et al., 1997; Edwards et al., 1998; Portella and Hewett, 1998). These methods focused on the generation of a single grid for the entire flow domain. However, it is unlikely that a single grid will be optimal for the whole reservoir. A modular gridding approach, in which the reservoir is decomposed into a set of subdomains and a grid is generated for each, permits much greater flexibility. Such an approach allows for the use of a grid optimized for each reservoir region.

The focus of this chapter is on the description of modular flow-based grids that can be generated locally and independently. A single local flow response is used to define mean flow streamlines that are naturally clustered in high velocity regions of a module. The streamline properties are used to automatically concentrate fine grid cells in important flow regions. The flow-based

grids and conventional Cartesian grid formulations are compared in terms of performance on coarsened reservoir models. Results computed with the flow-based grids demonstrate a significant improvement in accuracy when compared with conventional Cartesian grids of similar resolution.

This chapter proceeds as follows. We first provide an overview of our modular grid generation approach. We then discuss our procedure, based on the use of streamlines and equipotentials, for the generation of the grid within any module. Issues related to the upscaling of fine grid properties and the recombination of modular grids for global flow simulation are considered next. Then, we present detailed flow results for three example problems that illustrate the potential benefits of our overall grid generation capability. Conclusions and indications of future research directions are then discussed.

3.2 Modular versus Global Grid Generation

Generation of a grid via a single technique such as an algebraic method (Gordon and Hall, 1973), Thompson mapping (Thompson et al., 1985), conformal mapping or even streamlines and equipotentials may prove to be adequate for certain reservoir types. For example, if a given well pattern is to remain fixed throughout the period of simulation, then a streamline-based grid generated using the specific well conditions may lead to an appropriate “flow-based” grid. However, the actual streamlines of the problem will change with time and thus the selection of a single static grid that is the most suitable for the entire flow simulation will inevitably require some tuning. In addition to problem tuning, such a specific grid may have to be changed globally for every new well configuration encountered. While we do not rule out these possibilities, our interest is in methods of grid generation that will allow greater flexibility for general-purpose reservoir simulation.

Rather than generate problem specific grids, the approach adopted here is to build the reservoir grid from a set of modular grids, where each local grid is appropriately chosen according to the local geometry, geology and/or *mean* flow conditions. Accordingly the reservoir is decomposed into a set of subdomains. Let us assume that for each well in the reservoir an appropriate *well* module can be defined which resolves the local well geometry and near-well heterogeneity. It then remains to define the grids for subdomains without wells. We call these latter grids *field* modules. Since the well modules will occupy a relatively small volume of the reservoir, the

larger component of grid generation will be performed using *field* modules. The advantages of this strategy are that the *field* modules can be generated independently using the most appropriate method locally. Changes in the global grid due to changing boundary conditions could be accommodated through module replacement in a dynamic setting. This will be investigated in future work.

In our quest for a generally applicable *field* module, let us suppose that we begin with a homogeneous brick shaped subdomain. The size and relative scale of the subdomain will depend on how the reservoir is locally subdivided. In the absence of heterogeneity, a local Cartesian grid would serve as an optimal generic field module that can be connected to other field or well modules. However, if any form of heterogeneity is present in the region, then a key requirement of an optimal *field* module is that the grid be dense in high flow regions so as to resolve connected flow paths in the reservoir.

The flow-based gridding method developed here exploits the natural clustering properties of streamlines in high flow regions and along connected flow paths of the reservoir. In this chapter we focus on the most fundamental method and develop grids directly from streamlines and equipotentials. The method involves a two step grid generation process. An initial sub-division of the reservoir is performed and subdomains are defined. This can be accomplished through use of flow-based information from a preliminary coarse grid calculation (Li, 1999) or in some other more approximate manner. This step will provide an initial estimate of connectivity. *Well* and *field* modules are then defined by appropriate gridding techniques. The two step procedure can be iterative depending upon the quality of the initial grid and domain subdivision. Various types of *field* modules can also be used. For example, specific features such as faults and pinchouts may require locally unstructured grids. Algebraic mappings, or Thompson mappings, may be used in other regions where geometry is well defined such as at internal or external reservoir boundaries. Flow-based modules will be of prime importance for generally heterogeneous regions.

3.3 K-Orthogonal Streamline-Equipotential Grids

Grids that are generated by using streamlines and equipotential lines have certain beneficial properties for discretization in reservoir simulation, which are illustrated with respect to the sin-

gle phase pressure equation formulated in general curvilinear coordinates. The pressure equation for a control volume with surface $\partial\Omega$ can be written in the form

$$\oint_{\partial\Omega} -(K\nabla\phi) \cdot \hat{n} ds = M = \sum_{i=1}^{N_d} \Delta F_i, \quad (3-1)$$

where ϕ is the pressure, K is the permeability tensor, N_d is the spatial dimension and M includes gravity and prescribed flux terms. The right hand side term is equal to the sum of the normal flux differences in each curvilinear coordinate direction. The outward normal flux F_i is given by:

$$F_i = - \int \sum_{j=1}^{N_d} T_{ij} \phi_{\xi_j} d\Gamma, \quad (3-2)$$

where $\phi_{\xi_j} = \partial\phi/\partial\xi_j$ and the general tensor T is defined as,

$$T = |J| J^{-1} K J^{-T}. \quad (3-3)$$

Here, $J_{ij} = \partial x_i / \partial \xi_j$ is an element of the Jacobian of the curvilinear coordinate transformation and $|J|$ denotes its determinant.

Edwards (1999) presented an analysis of the use of streamline grids for flow problems characterized by full tensor permeability fields. There it is proven that the general tensor field corresponding to *any* streamline-equipotential coordinate system is always strictly diagonal. Moreover this result holds for any diagonal or full Cartesian permeability tensor field. In practice the streamlines and potential field are generated using fine scale information while the streamline-equipotential grid is generated at a coarser grid level. Consequently the diagonal tensor property will be approximate at any finite grid level and this property is referred to as quasi K-orthogonality. In this case the tensor T reduces to

$$T^D = \text{diag}(|J| J^{-1} K J^{-T}), \quad (3-4)$$

where T^D denotes the diagonal tensor. The flux in this case reduces to $F_i = - \int T_i^D \phi_{\xi_i} d\Gamma$.

This observation suggests that, when using a streamline-potential grid, a consistent flux approximation can be defined by the product of the local leading diagonal component of the general tensor matrix and the standard two-point pressure difference in the corresponding K-orthogonal coordinate direction. This has also been verified by simulation (Edwards, 1999). Consequently, standard simulators can be applied to arbitrary full tensor field problems provided

that a suitable streamline - equipotential grid is generated. Thus K-orthogonality together with the natural clustering properties of streamlines provide strong motivations for developing flow modules based directly on local streamline information.

3.4 Generation of Flow Response Based Modules

The flow-based modules presented here are generated from a single-phase flow calculation. The choice of boundary conditions can greatly influence the grid geometry. In terms of defining a grid *field* module, while we wish to exploit the attractive properties of a flow-based grid, the most generic module will result from the use of the least problem dependent boundary conditions. In this work, uniform influx is imposed on one surface of the brick subdomain and fixed pressure is imposed on the opposite surface. No-flow conditions are imposed on the four remaining reservoir faces. These boundary conditions provide a set of streamlines corresponding to a uniform inlet. Note that the reservoir subdomain will collapse to a Cartesian grid if no heterogeneity is present.

Because the choice of boundary conditions is not unique, another possibility for defining a generic flow module is to generate the local grid by performing three uniform influx simulations (one in each coordinate direction) and then establishing a mean grid based on this flow information. The effects of module boundary conditions are still under investigation. By choosing appropriate mean flow conditions we aim to avoid tuning the grid to specific boundary conditions. It must be stressed that the streamlines and equipotentials used to build a generic *field* module can be quite independent from the streamlines and equipotentials that result from a specific problem. However, they still provide a simple method of generating a local grid with key properties with respect to flow resolution that can be difficult to obtain with other grid generation techniques.

The flow-based grids are built using natural streamline-equipotential coordinate systems that result from mean flow responses in heterogeneous media. The specific steps involved in generating a three-dimensional flow module are given below:

- (a) First the three dimensional module geometry and the initial fine scale grid are defined, a single phase flow calculation is performed by solving the pressure equation subject to uniform inflow and outflow boundary conditions, and streamlines are traced using the fine grid

velocity field with the method of Thiele et al. (1996).

- (b) The nodes along the streamlines are coarsened according to their local density, for a specified level of resolution on the coarse grid. As seen in Fig. 3-1a, starting from a three dimensional trajectory, the curvilinear coordinates of each point defining the curve are computed. According to the level of resolution needed, groups of points are formed and their barycenters are computed. The new reduced set of points is then mapped onto the initial three dimensional trajectory.
- (c) A structured (layered) set of streamlines is defined over the injection face (Fig. 3-1b), where each layer contains the same number of streamlines. The streamlines are selected within each layer according to streamline density for a specified level of resolution on the coarse grid (Fig. 3-1b) using the same method as in (b).
- (d) Layers of streamlines are then recomputed according to layer density for a specified level of resolution, using again the same method as in (b).
- (e) The pressure field is projected onto the selected streamlines at logical uniform intervals via trilinear interpolation, where the interpolated pressure is a function of the corresponding eight cell centered pressures, as indicated in Fig. 3-1c.
- (f) For each layer, equipotential line interpolation is performed across the selected streamlines, resulting in a set of logically Cartesian streamline-equipotential-line surfaces. Equipotential surfaces are then defined by joining the surface layers to form the three dimensional grid. The grid generation process is illustrated in Fig. 3-1d.
- (g) The fine grid permeabilities are then upscaled onto the new coarse grid. This important step remains to be fully developed and is discussed further below.

Note that this method of grid generation requires the solution of one partial differential equation in three-dimensions, while comparable methods require the solution of three partial differential equations (e.g., Thompson et al., 1985).

(a)

(b)

(c)

(d)

Figure 3-1: Grid generation in three dimensions: (a) coarsening of nodes along the streamlines, (b) coarsening of streamlines within layers, (c) trilinear interpolation of pressure field onto streamlines, (d) grid block construction by equipotential line interpolation

3.5 Upscaling and Grid Smoothing

Various procedures for computing the upscaled grid block permeability tensors can be employed. As demonstrated in the Results section below, coarse scale flow results may be sensitive to the choice of method. This sensitivity emphasizes the important link between upscaling and grid generation. Currently, flow response based upscaling algorithms, even with simple pressure-flux or periodic boundary conditions, cannot be applied directly. The main difficulty here is that the flow-based coarse scale grid does not generally conform to the underlying fine scale grid. Therefore, a consistent flow response upscaling technique to determine the upscaled tensor involves development of appropriate fine to coarse grid operators. Such procedures have recently been developed for two dimensional systems (He et al., 2001; Wen et al., 2000); the method of He et al. (2001) is presented in Chapter 6 of this report. We anticipate further scope for improvement in flow resolution when a more robust and general coarse grid to fine grid three-dimensional upscaling algorithm is available. In addition, for K-orthogonal grids, no-flow boundary conditions apply since cross flow is absent (Edwards, 1999).

In our current work, upscaled permeabilities are computed using the commercial package FloGrid (Schlumberger, 1999b). These permeabilities are generated using simple averages, or

combinations of averages, of the underlying fine scale permeabilities. In the results below, we use a combined arithmetic - harmonic averaging unless otherwise specified. For the Cartesian coarse grid results, flow-based upscaled permeabilities computed using pressure - flux boundary conditions are applied. In all cases the fine scale relative permeabilities are used directly on the coarse flow-based grid; i.e., pseudo relative permeabilities are not introduced.

As described above, our procedure allows for a three dimensional domain decomposition to be performed prior to generating the grid for each module. In this case each *field* module is generated independently from a local single-phase flow response. The global grid is then built by blending the modules together. This is achieved by local Laplacian smoothing of grid nodes that lie on common surfaces that are shared by neighboring modules. The remaining majority of grid nodes are fixed, although smoothing of some interior regions can be performed when necessary. Interior smoothing may be beneficial when the streamlines are densely concentrated in certain regions of a module and very sparse elsewhere. Generation of flow-based grids with smoothing strategies will be the subject of future work. In the results presented below, interior smoothing is not applied away from module interfaces.

3.6 Flow Simulation Results

Our simulation results are for two-phase oil – water systems. Quadratic relative permeabilities are used and capillary pressure is neglected. The streamline grids are generated from a flow response corresponding to uniform influx and constant pressure boundary conditions, imposed on the left and right hand sides of the system respectively. In the first two examples the streamlines are generated globally (i.e., a single grid is used) while in the third example a modular approach is applied. All simulation results were generated using the Eclipse (Schlumberger, 1999a) reservoir simulator.

The first case involves a heterogeneous, two dimensional, channel system (Fig. 3-2a) characterized by a bimodal permeability distribution (Fig. 3-2b). This system, of dimensions 100×130 , was generated using sequential Gaussian simulation (Mao and Journel, 1999). The coarse (8×9) Cartesian grid and corresponding grid block permeabilities are shown in Fig. 3-2c. The flow-based grid and its associated permeabilities, shown in Figs. 3-2d and 3-2e, clearly demon-

strate the ability of the flow-based technique to concentrate grid blocks in high flow regions (i.e., in the high permeability channels).

In the flow simulations, flow is driven by a single injector and producer located in the high permeability channel, as indicated in the figures. These locations are quite distinct from the boundary conditions used to generate the streamline grid. Flow simulation results are shown in Fig. 3-2f, where we plot watercut at the production well as a function of time for the reference fine scale grid (100×130), the 8×9 streamline grid and the 8×9 coarse Cartesian grid. The comparison clearly shows the improvement offered by the flow-based grid over a standard Cartesian grid of identical resolution.

(a)

(b)

(c)

(d)

Figure 3-2: Results for two dimensional system: (a) permeability field for fluvial channel system (log scale), (b) bimodal permeability distribution, (c) upscaled Cartesian permeability field (log scale), (d) flow-based grid showing wells, (e) upscaled permeability field on flow-based grid (log scale), (f) results for watercut

Our second example involves a three-dimensional channeled system, (of dimensions $80 \times 80 \times 20$). The system, containing a group of heterogeneous channels, is shown in Fig. 3-3a. The mean permeability contrast between the channel sand and the background shale is 2000:1. The three dimensional flow-based grid is shown in Fig. 3-3b. The correspondence between the grid and the underlying permeability field may be seen more clearly in Fig. 3-3c, where two layers of the grid, corresponding to the region extracted in Fig. 3-3a, are presented. The upscaled permeability field for these two layers is shown in Fig. 3-3d.

The boundary conditions for the two-phase flow simulations are the same as those used to generate the streamline grid; i.e., water is injected uniformly over the left face of the model and pressure is specified on the right hand face. Flow simulation water cuts computed using the reference $80 \times 80 \times 20$ Cartesian fine grid, $16 \times 16 \times 12$ and $8 \times 8 \times 8$ mean streamline grids, and a $16 \times 16 \times 12$ coarse Cartesian grid are shown in Fig. 3-3e. As we saw in the previous example, the benefits of using flow-based grids are clearly demonstrated. The $8 \times 8 \times 8$ streamline grid provides more accurate results, relative to the reference case, than does the more finely resolved $16 \times 16 \times 12$ coarse Cartesian grid. The $16 \times 16 \times 12$ streamline grid simulation shows even further improvement, indicating a good step towards solution convergence.

The sensitivity of the simulation results to the choice of upscaling technique is illustrated in Fig. 3-3f. Here, results are presented for the $16 \times 16 \times 12$ streamline grid with grid block permeabilities computed using either a combination of arithmetic and harmonic averages (as in the results of Fig. 3-3e) or a volume weighted geometric average. The results for these two simulations are somewhat different, particularly around the time of water breakthrough. This illustrates the importance of permeability upscaling and demonstrates the need for the development of accurate techniques for the calculation of grid block permeabilities for use in flow-based models.

(d)

(e)

(f)

Figure 3-3: Results for three dimensional system: (a) high permeability heterogeneous channel sands, (b) flow-based grid – inside view, (c) flow-based grid – two layers, (d) two layers with upscaled permeability, (e) water cut – flow-based grid convergence, (f) effects of upscaling method on water cut results

The final example involves the same three dimensional system as in the previous case. Now, however, the global $16 \times 16 \times 12$ reservoir grid is built from a set of four *field* modules, as illustrated in Fig. 3-4a. The grid generation procedure used is that described in section 3.4 above. Layers of the upscaled permeability field corresponding to the sub-domains of Fig. 3-4a are shown in Fig. 3-4b. The global flow-based grid is depicted in Fig. 3-4c. Flow simulation results using this modular grid, shown in Fig. 3-4d, are clearly superior to those obtained from the coarse Cartesian grid and are very close to those obtained with the global streamline grid. This example serves to demonstrate that modular flow-based grids can be generated independently and recomposed to form a global grid that successfully captures the primary flow paths of the reservoir.

Due to grid coarsening and upscaling, the grids employed here are only quasi K-orthogonal (Edwards et al., 1998) as a result of numerical approximations. However, the comparison with the fine grid simulations indicates that major benefits are still obtained when using the mean flow-based grids compared to corresponding uniform Cartesian grids of identical resolution.

(a)

(b)

(c)

(d)

Figure 3-4: Results for modular three dimensional system: (a) domain decomposition – four field modules, (b) layers of field modules – upscaled permeability, (c) global flow-based grid composed of four field modules, (d) water cut comparison – results for modular and standard flow-based grids

3.7 Conclusions

In this chapter, a strategy for modular grid generation in two and three dimensions is presented. The basic technique entails the use of streamline and equipotential flow information generated from the solution of a single-phase flow problem on the fine scale (geostatistical) model. The resulting flow-based information is used to generate grids with a graded distribution of nodes concentrated in critical high flow regions of the reservoir. In addition, because the resulting grids are derived from streamlines and equipotential surfaces that are quasi-K-orthogonal at the coarse grid level, the general tensor field is approximately diagonal throughout the domain. The modularity of the method allows for the generation of different grids in different regions of the reservoir. These grids are subsequently combined to form the global reservoir grid.

Simulation results using the new procedures demonstrate the significant improvement in accuracy attainable over that of simpler (uniform Cartesian coarsening) approaches. Enhanced accuracy in terms of water-cut is observed over the entire course of the simulation runs, though improvements in the accuracy of breakthrough time predictions are especially apparent. Some sensitivity to the choice of upscaling technique is observed in the coarse scale results. This indicates that future work will need to focus on the development of accurate methods for the calculation of coarse scale grid block permeabilities for use with three dimensional flow-based grid generation techniques. Work in this direction is currently underway.

3.8 References

- Durlofsky, L.J., Jones, R.C. and Milliken, W.J.: "A Nonuniform Coarsening Approach for the Scale-Up of Displacement Processes in Heterogeneous Porous Media," *Adv. Water Resour.*, **20**, 335-347, 1997.
- Edwards, M.G.: "A Higher Order Godunov Scheme Coupled with Dynamic Local Grid Refinement for Flow in a Porous Medium," *Comput. Methods. Appl. Mech. Engrg.*, **131**, 287-308, 1996.
- Edwards, M.G.: "Split Tensor Discretization Operators Coupled with Quasi K-Orthogonal Grids," paper SPE 51903 presented at the SPE Symposium on Reservoir Simulation, Houston, Feb. 14-17, 1999.

- Edwards, M.G., Agut, R. and Aziz, K.: "Quasi K-Orthogonal Streamline Grids: Gridding and Discretization," paper SPE 49072 presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Sept. 27-30, 1998.
- Gordon, W.J. and Hall, C.A.: "Construction of Curvilinear Coordinate Systems and Applications to Mesh Generation," *Int. J. for Num. Methods in Eng.*, **7**, 461-477, 1973.
- He, C., Edwards, M.G. and Durlofsky, L.J.: "Numerical Calculation of Equivalent Cell Permeability Tensors for General Quadrilateral Control Volumes," in review for *Comp. Geosciences*, 2001.
- Li, B.: "Modular Streamline Based Grids for Reservoir Simulation," Master's report, Stanford University, 1999.
- Mao, S. and Journel, A.G.: "Generation of a Reference Petrophysical / Seismic Data Set: The Stanford IV Reservoir," Stanford Center for Reservoir Forecasting Report, May 1999.
- Portella, R.C.M. and Hewett, T.A.: "Upscaling, Gridding, and Simulation using Streamtubes," paper SPE 49071 presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Sept. 28-30, 1998.
- Schlumberger Geoquest: Eclipse Reservoir Simulator, 99A Release, 1999a.
- Schlumberger Geoquest: FloGrid, 1999b.
- Thiele, M.R., Batycky, R.P., Blunt, M.J. and Orr, F.M.: "Simulating Flow in Heterogeneous Systems using Streamtubes and Streamlines," *SPE Res. Eng.*, **11**, 5-12, 1996.
- Thompson, J.F., Warsi, Z. and Mastin, C.W.: *Numerical Grid Generation – Foundations and Applications*, Elsevier Science Publishing Co., New York, 1985.
- Tran, T.T.: "Stochastic Simulation of Permeability Fields and their Scale Up for Flow Modeling," Ph.D. thesis, Stanford University, 1995.
- Wen, X.H., Durlofsky, L.J., Lee, S.H. and Edwards, M.G.: "Full Tensor Upscaling of Geologically Complex Reservoir Descriptions, paper SPE 62928 presented at the SPE Annual Technical Conference and Exhibition, Dallas, Oct. 1-4, 2000.

Part II. Coupling of the Reservoir and Non-Conventional Wells in Simulators

An important aspect of our research is the development of accurate and computationally efficient models for wellbore flow that can be used to couple wellbore hydraulics to reservoir flow in simulators. The drift-flux model is a two-phase flow model that is commonly used in this context. Here we describe our work on the determination of accurate drift-flux model parameters from experimental data. The enhanced formulation presented here is applicable for use in the well model used in reservoir simulators.

4 Investigation of Drift-Flux Model Parameters

by Yuguang Chen, Louis J. Durlofsky and Khalid Aziz

4.1 Introduction

The two-phase flow of gas and liquids in wells and pipelines is a problem of great importance in the petroleum industry. The presence of two flowing phases in the wellbore can alter the pressure profile significantly from that observed for single phase flow. In the case of long horizontal or deviated wells, wellbore pressure losses can have a significant effect on reservoir productivity. Thus, it is important to be able to predict the characteristics of two-phase wellbore flows.

The behavior of gas-liquid systems is difficult to predict because the interface between the two phases can be distributed in many configurations. This phenomenon is called flow pattern, which is a fundamental feature of two-phase flows. In single-phase flow in pipes, the design parameters such as pressure drop can be calculated in a relatively straightforward way. However, the existence of a second phase presents a difficult challenge in the understanding and modeling of the flow system. The hydrodynamics of the flow, as well as the flow mechanisms, change significantly from one flow pattern to another. For instance, it has been demonstrated (Cheremisinoff, 1986) that for similar flow conditions, slug flow and wavy flow may result in a difference in pressure drop of a factor of two. Some heat transfer parameters estimated using

correlations for the stratified flow pattern change by several orders of magnitude from those estimated from correlations for the annular flow pattern.

In petroleum engineering applications, the three most important hydrodynamic features are the flow pattern, the holdup of the two phases, and the pressure drop. In order to estimate accurately the pressure drop and holdup, it is necessary to know the actual flow pattern under the specific flow conditions. Mechanistic models (e.g., Barnea, 1987; Chen et al., 1997; Petalas and Aziz, 1997, 1998; Kaya et al., 1999; Gomez et al., 2000) can provide this information, though the procedure for determining flow patterns is nontrivial and can be time consuming computationally. Further, the discontinuities in pressure drop and holdup due to the shift from one flow regime to another may give rise to convergence problems if a mechanistic wellbore flow model is coupled with a reservoir simulator. Thus, these models are probably not the most appropriate for use within a reservoir simulator. Rather, simplified models that are better suited for use with reservoir simulators are required.

For the coupled simulation of flow in the reservoir and the wellbore, the flow model is required to be efficient, continuous and differentiable (Schlumberger GeoQuest, 2000). The drift flux model is well suited for this purpose. With two parameters (distribution/profile parameter C_0 and drift velocity V_d), holdup can be calculated from the superficial velocities.

The drift flux model was first proposed by Zuber & Findlay (1965). Usually, this model is applied to vertical dispersed systems. In the Petalas & Aziz (1997, 1998) mechanistic model, in the intermittent flow, dispersed bubble flow and bubble flow regimes, the holdup is calculated by the drift flux model and then the pressure drop is obtained using a homogeneous model. Some effort has also been put into the investigation of the two parameters C_0 and V_d . Petalas & Aziz (1997) correlated the profile parameter and drift velocity with the liquid Reynolds number using the data points in the above three flow patterns. Mishima & Ishii (1984) related the profile parameter with fluid densities. This expression was used by Ouyang (1998) in a homogeneous model with slip for gas-liquid wellbore flow. Similarly, Eclipse (Schlumberger GeoQuest, 2000) uses the drift flux model in calculations for multi-segment wells. The correlations in Eclipse were synthesized from several other published correlations. They depend on fluid properties, gas volume fraction, mixture velocity and pipe inclination angle.

In this chapter, the drift-flux model (DFM) is applied to different flow patterns. Based on our observations, we model the parameters C_0 and V_d in DFM as linear functions of the gas volume fraction α_G . A method is proposed to determine these two parameters by matching the experimental α_G and the α_G calculated from DFM as closely as possible. The physical meanings of C_0 and V_d are illustrated by analyzing the resulting correlations. Finally, comparisons between the drift-flux model correlations in Eclipse (Schlumberger GeoQuest, 2000) and the experimental data are presented.

4.2 Basic Definitions for Two Phase Pipe Flow

The superficial velocities of the liquid and gas phases (V_{SL} and V_{SG}) are defined as the volumetric flow rate for the phase divided by the pipe cross sectional area:

$$V_{SL} = \frac{Q_L}{A}, \quad V_{SG} = \frac{Q_G}{A}, \quad (4-1)$$

where Q_L and Q_G are the volumetric flow rate of liquid and gas respectively and A is the pipe cross sectional area. The mixture velocity is given by the sum of the gas and liquid superficial velocities:

$$V_M = V_{SL} + V_{SG}. \quad (4-2)$$

The input volume fractions of the liquid and gas phases (C_L and C_G) are defined as:

$$C_L = \frac{Q_L}{Q_L + Q_G} = \frac{V_{SL}}{V_M}, \quad (4-3)$$

$$C_G = \frac{Q_G}{Q_L + Q_G} = \frac{V_{SG}}{V_M}. \quad (4-4)$$

By definition the sum of the liquid and gas volume fractions is equal to one.

The characteristic of two-phase flow is the simultaneous flow of two phases of different density and viscosity. Usually in horizontal and uphill flows, the less dense and/or less viscous phase tends to flow at a faster velocity. In gas-liquid flow, gas moves much faster than liquid except in downward flow. The difference in the in situ average velocities between the two phases results in a very important phenomenon --- the “slip” of one phase relative to the other, or the

“holdup” of one phase relative to the other (Govier & Aziz, 1972). This makes the in situ volume fractions different than the input volume fractions. Although “holdup” can be defined as the fraction of the pipe volume occupied by a given phase, holdup is usually defined as the in situ liquid volume fraction, while the term “void fraction” is used for the in situ gas volume fraction.

Let the cross sectional area occupied by liquid be A_L ; the remaining area A_G is occupied by gas. The liquid holdup and gas volume fraction are defined as:

$$\alpha_L = \frac{A_L}{A}, \quad \alpha_G = \frac{A_G}{A}. \quad (4-5)$$

After the in situ volume fraction is known, we can calculate the average (in situ) velocity for each phase:

$$V_L = \frac{Q_L}{A_L} = \frac{V_{SL}}{\alpha_L}, \quad (4-6)$$

$$V_G = \frac{Q_G}{A_G} = \frac{V_{SG}}{\alpha_G}. \quad (4-7)$$

These are the true average velocities of the liquid and gas phases, which are larger than the superficial velocities.

Fluid properties (density, viscosity and interfacial tension) for each phase and geometric parameters such as the pipe internal diameter and pipe inclination angle also have an influence on the performance of the system. In this work, the pipe inclination angle θ is measured from the horizontal except when otherwise noted.

4.3 Drift-Flux Model Parameters

The drift-flux model proposed by Zuber & Findlay (1965) can be used to calculate the gas volume fraction and interpret holdup data. It correlates the actual gas velocity V_G and the mixture velocity V_M using two parameters C_0 and V_d :

$$V_G = \frac{V_{SG}}{\alpha_G} = C_0 V_M + V_d, \quad (4-8)$$

Figure 4-1: Schematic of velocity and concentration profiles

where V_M is the mixture velocity as defined in Eq. (4-2). C_0 is referred to as the distribution parameter or profile parameter. It accounts for the effects of the non-uniform distribution of both velocity and concentration profiles (see Fig. 4-1 for typical gas concentration and velocity distributions). If the two phases are uniformly mixed, the concentration profile will be flat and C_0 should be equal to one. V_d is called the drift velocity of gas, and accounts for the local relative velocity between the two phases. If the liquid is stationary, V_d corresponds to the gas rise velocity in the stagnant liquid.

With the two parameters and superficial velocities, the in situ gas volume fraction can be calculated. The accuracy of the predicted α_G depends on the use of appropriate values for the C_0 and V_d parameters.

4.4 Drift-Flux Model in Different Flow Patterns

Traditionally, the drift-flux model is used most widely for vertical dispersed systems. Eq. (4-8) is derived from the continuity equation in dispersed systems (Govier & Aziz, 1972). Nonetheless, the linear relationship between V_G and V_M has been confirmed empirically for flow regimes other than dispersed flow. These even include separated horizontal flows (Franca & Lahey, 1992).

In Figs. 4-2 and 4-3, we plot V_G vs. V_M in different flow patterns for both horizontal and vertical flows. Using linear regression, we determine the two parameters C_0 and V_d . Very high degrees of correlation for V_G and V_M are observed for all the flow conditions (values of R^2 greater than 0.98 in all cases, where R^2 is the square of correlation coefficient).

Since different flow mechanisms operate in different flow patterns, the DFM parameters C_0 and V_d should depend on the flow regimes and flow orientations. If we consider vertical flow, we note that the value of C_0 is close to 1 in annular-mist flow, while it is about 1.16 in slug flow. This behavior can be explained through consideration of the concentration profiles in annular-mist and slug flows. In annular-mist flow, although there are a few liquid droplets entrained in the gas core, the overall gas distribution is fairly uniform. Thus, $C_0 \sim 1$. However, in slug flow where Taylor bubbles and liquid slugs appear alternatively, the non-uniform effects are much stronger, which gives rise to $C_0 > 1$.

Figs. 4-4 and 4-5 summarize the values of C_0 and V_d obtained in various flow patterns in vertical flow. We calculate the average gas volume fraction in each flow pattern and use it for the presentation of the results. The lowest gas volume fraction occurs in the elongated bubble flow. The gas volume fraction increases in slug flow as the entrained gas bubbles can exist in the liquid slug. The highest void fraction is in annular-mist flow because of the existence of the gas core. There are significant differences in the values of C_0 and V_d among the various flow patterns.

The relation between flow pattern information and volume fraction can be demonstrated more clearly via flow pattern maps. Instead of using V_{SL} vs. V_{SG} (as is commonly done), we use the coordinates of V_M vs. α_G in Figs. 4-6 and 4-7 to represent the information on flow patterns. Different shades represent different flow patterns. For vertical flow, at low α_G and V_M , elongated bubble flow is observed. As α_G and V_M increase, slug and churn flows occur, and finally annular-mist flow is achieved. In horizontal flow, stratified flow is also associated with high gas volume

fractions (Fig. 4-6). However, it can be differentiated from the intermittent and annular-mist flows by its lower flow rate. The flow regimes in vertical flow are relatively simple due to the absence of stratified flow. Therefore, α_G itself can provide us with reasonably accurate information on flow patterns (Fig. 4-7).

We see from Figs. 4-2 to 4-7 that flow patterns have a strong influence on the DFM parameters. Thus it is not adequate to take these two parameters as constants. We also see that α_G provides some indication of the flow pattern. In next section, we will describe our estimation of C_0 and V_d based on α_G .

4.5 Method for Parameter Determination

4.5.1 Objective Function using α_G . With the DFM parameters we obtained by linear regression, we can calculate the in situ gas volume fraction:

$$\alpha_G = \frac{V_{SG}}{C_0 V_M + V_d}. \quad (4-9)$$

Fig. 4-8 (a) shows the scatter plot between the predicted α_G and the experimental α_G^* . Different shades represent data points in different flow patterns, where the shading is the same as that in Fig. 4-7. Fig. 4-8(b) displays the comparison between the experimental V_G and predicted V_G . Compared to the excellent agreement of V_G in Fig. 4-8(b), the prediction for α_G is much less accurate.

The aim of the procedure applied above is to determine C_0 and V_d by minimizing the error of the following objective function:

$$E_{V_G} = \sum_{i=1}^m \left[\left(\frac{V_{SG}}{\alpha_G} \right)_i - (C_0 V_{Mi} + V_d) \right]^2, \quad (4-10)$$

* In a scatter plot, the correlation coefficient ρ is used to quantify the linear dependence between two variables. For the two variables X and Y , ρ is defined as: $\rho = \sigma_{XY} / (\sigma_X \sigma_Y)$, where σ_{XY} is the covariance between X and Y , σ_X and σ_Y the standard deviation of X and Y respectively. The previously used R^2 in the $V_G \sim V_M$ plot is given by $R^2 = \rho^2$.

Figure 4-2: Drift-flux model in horizontal flow (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

Figure 4-3: Drift-flux model in vertical flow (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

Figure 4-4: Values of C_0 in different flow patterns (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

Figure 4-5: Values of V_d in different flow patterns (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

Figure 4-6: Flow pattern map in coordinates of V_M vs. α_G (Data: Spedding & Nguyen (1976), air-water system, horizontal flow, $D=1.79$ inch)

Figure 4-7: Flow pattern map in coordinates of V_M vs. α_G (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

where m is the total number of experimental data points. Since our objective here is to predict the in situ gas volume fraction as accurately as possible, a more appropriate way of determining C_0 and V_d is to directly minimize the error between the measured α_G and the estimated α_G :

$$E_{\alpha_G} = \sum_{i=1}^m \left[\alpha_{Gi} - \left(\frac{V_{SGi}}{C_0 V_{Mi} + V_d} \right) \right]^2. \quad (4-11)$$

Minimization of Eq. (4-11) is a nonlinear least square problem. A Gauss Newton algorithm (Li *et al.*, 1995) is used to solve this system.

Using C_0 and V_d determined from Eq. (4-11), the comparisons between the predicted and measured values for both α_G and V_G are presented in Fig. 4-9. Significant improvement is obtained for the calculation of α_G except at very high α_G . Note that from the two methods (Eqs. (4-10) and (4-11)), we get different values for C_0 and V_d . However, in Figs. 4-8(b) and 4-9(b), we see that the accuracy of V_G for these two methods is the same, and both have very high correlation coefficients. The discrepancies between the predictions for α_G and V_G indicate that V_G is less sensitive to the values of the DFM parameters. In other words, the actual gas velocity is not a good indicator of how well the drift-flux model works for the prediction of the in situ gas volume fraction.

Figure 4-8: Prediction results using E_{V_G} (Data: SU66, Govier *et al.* (1957), air-water system, vertical flow, $D=1.02$ inch)

Figure 4-9: Prediction results using E_{α_G} (Data: SU66, Govier *et al.* (1957), air-water system, vertical flow, $D=1.02$ inch)

The impact of flow patterns on the values of C_0 and V_d was shown in section 4.2. Instead of calculating these two parameters using the data over the entire range of interest, we can apply Eq. (4-10) and Eq. (4-11) to each flow pattern. Fig. 4-10 displays this result using the objective function E_{V_G} , while Fig. 4-11 is for the objective function E_{α_G} . Again, a better match is obtained when E_{α_G} is used (Figs. 4-10(a) and 4-11(a)). One interesting observation is that when the flow pattern information is taken into account, E_{V_G} can give results comparable to those using E_{α_G} (Figs. 4-10(b) and 4-11(b)). However, it should be kept in mind that we generally do not know flow pattern information in advance. Thus we conclude that E_{α_G} is the better way of determining C_0 and V_d . In the next section, α_G will be introduced to indicate the flow patterns and E_{α_G} will be used as the objective function.

Figure 4-10: Prediction results using E_{V_G} (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

Figure 4-11: Prediction results using E_{α_G} (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

4.5.2 Incorporation of α_G into Correlations of C_0 and V_d . We represent C_0 and V_d as functions of α_G . As a first approximation, the following linear functions are considered:

$$\begin{aligned} C_0 &= a\alpha_G + b \\ V_d &= c\alpha_G + d \end{aligned} \quad (4-12)$$

where a , b , c and d are constants. Substituting Eq. (4-12) into Eq. (4-11), we get the following objective function:

$$E_{\alpha_G} = \sum_{i=1}^m [\alpha_{Gi} - \alpha_{Gi}^*]^2, \quad (4-13)$$

where

$$\alpha_{Gi}^* = \frac{V_{SGi}}{(a\alpha_{Gi}^* + b)V_{Mi} + (c\alpha_{Gi}^* + d)}. \quad (4-14)$$

We note that Eq. (4-14) is implicit in α_G . However, since a linear expression for C_0 and V_d is assumed, the estimated α_G can be expressed explicitly. Therefore, the solution procedure for Eq. (4-13) is still a standard nonlinear least squares problem, and the same Gauss Newton algorithm applied in the previous section is used.

The results using this method are shown in Figs. 4-12 and 4-13. Compared to Fig. 4-9 (a), we get a better prediction for α_G in Fig. 4-12, especially in the annular-mist flow regime (α_G near 1). For another data set, the same observation can be made (compare Figs. 4-11 (a) and 4-13), though the improvement here is not as significant as in the previous case.

We have shown that when the flow pattern information is considered, E_{V_G} and E_{α_G} can provide us with equivalent predictions (Figs. 4-10(b) and 4-11(b)). So, it is worthwhile to consider using Eq. (4-12) in Eq. (4-10), to account for the effects of α_G when using E_{V_G} as the objective function. The advantage of E_{V_G} is the linearity in the resulting least square problem and the explicit expression for the estimated V_G . This will allow for the use of simpler algorithms when a more complicated form for C_0 and V_d is used. Unfortunately, as shown in Figure 4-14, the combination of the linear form for C_0 and V_d and the objective function of E_{V_G} does not give us results that are as accurate as those in Fig. 4-13. This demonstrates that there is a clear advantage in using E_{α_G} rather than E_{V_G} .

Figure 4-12: Prediction result using new approach (Data: SU66, Govier *et al.* (1957), air-water system, vertical flow, $D=1.02$ inch)

Figure 4-13: Prediction result using new approach (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

Figure 4-14: Prediction result using E_{V_G} and linear form of C_0 and V_d (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

4.6 Application of Proposed Method to Other Inclination Angles

So far, we have investigated only vertical flow. Earlier in this chapter, however, we showed that the drift-flux model is also applicable for horizontal flow. In this section, we will therefore apply the method described above to flows in other inclination angles. The objective is to assess the applicability of the drift-flux model to flow in other inclinations, as well as to test the robustness of the method we proposed for the determination of C_0 and V_d .

The results for upward and downward flows at various inclination angles (θ), using the data of Spedding & Nguyen (1976), are displayed in Fig. 4-15. For the upward flow ($\theta=70^\circ$, 45° and 21°), we obtain high correlation coefficients and the predictions are comparable to those for vertical flow. For near-horizontal and downward flows, the data show significant scatter. In downward flow, stratified flow takes the place of intermittent flow. Thus few data points are in the region of low gas volume fraction. The results for horizontal flow will be shown later.

Figure 4-15: Prediction results for other inclination angles (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

Fig. 4-16 shows results using other data sets in the Stanford Multiphase Flow Database. These data are all from the same source (Mukherjee, 1979), but the pipe diameter ranges from 1-2 inches. Again, the results for downward flow are not as good as those for upward flow.

Figure 4-16: Prediction results for other inclination angles (Data: SU175-SU198: Mukherjee, 1979)

We next consider horizontal flow. In Figs. 4-17(a) and 4-18(a), we show the results for the case with C_0 and V_d constant. Figs. 4-17(b) and 4-18(b) display the predictions for α_G when C_0 and V_d are estimated in each flow pattern. This data and the data for vertical flow presented in Figs. 4-10 and 4-11 are from the same researchers (Spedding & Nguyen, 1976). However, the results for horizontal flow are much less accurate than those for vertical flow. The data points in

the stratified flow regime show more scatter (in Fig. 4-17(a), they are toward the lower right of the figure), though a very high correlation between V_G and V_M was achieved in Fig. 4-2(a). When we apply the linear forms of C_0 and V_d to this data set, the nonlinear least squares algorithm may converge to unphysical (complex) values for C_0 and V_d . In this case, additional constraints must be introduced into the optimization procedure. This problem has been observed for some horizontal and downward flows, but not for upward flows.

Figure 4-17: Prediction results using E_{V_G} (Data: Spedding & Nguyen (1976), air-water system, horizontal flow, $D=1.79$ inch)

Figure 4-18: Prediction results using E_{α_g} (Data: Spedding & Nguyen (1976), air-water system, horizontal flow, $D=1.79$ inch)

Figure 4-19: Prediction result for horizontal flow (Data: Chen & Spedding (1979), air-water system, $D=1.79$ inch)

Figure 4-20: Prediction result for horizontal flow (Data: Franca & Lahey (1992), air-water system, $D=0.75$ inch)

For horizontal flow, good predictions using our method can be achieved for some data sets (see Figs. 4-19 and 4-20). However, the overall performance of the method for downward flow and horizontal flow is not as good as that achieved for vertical and upward flows. The determination of drift-flux parameters for downward and horizontal flow will require further investigation.

4.7 Discussion of Drift Velocity V_d

We proposed a method for the determination of drift-flux parameters that works well for upward flows. So far, we have only applied these parameters to the data set for which they were generated. To obtain a more general correlation, we need to apply the procedure to multiple data sets. However, before we move on to this step, it is useful to consider the physical meanings of C_0 and V_d . Is the linear model we assumed appropriate and adequate? Is the behavior we get here consistent with the physical observations? We now consider these issues.

4.7.1 Physical Meaning of C_0 and V_d . For most of the data sets, C_0 and V_d display behavior similar to that shown in Fig. 4-21. C_0 decreases to 1.0 as α_G approaches 1.0, while V_d increases with α_G . As we have explained, C_0 accounts for the effects of the non-uniform distribution of both velocity and concentration profiles. As α_G approaches 1.0, which means high gas volume fraction and high flow rates, the profiles tend to distribute uniformly, so $C_0 \sim 1$. When α_G approaches zero, non-uniform effects are stronger so C_0 deviates from one.

The quantity V_d accounts for the local relative velocity between the two phases. A limiting case can be thought of as a single gas bubble rising through a liquid. This single bubble rise velocity is also called terminal gas rise velocity, designated V_∞ . It can be calculated as follows, as determined from experimental results (Zuber & Findlay, 1965):

$$\text{in bubbly flow regime: } V_\infty = 1.53 \left[\frac{\sigma g (\rho_L - \rho_G)}{\rho_L^2} \right]^{\frac{1}{4}}, \quad (4-15)$$

$$\text{and in slug flow regime: } V_\infty = 0.35 \left[\frac{g (\rho_L - \rho_G) D}{\rho_L} \right]^{\frac{1}{2}}. \quad (4-16)$$

For the air-water system of Fig. 4-21, this velocity is 0.25m/s (Eq. (4-15)) or 0.23 m/s (Eq. (4-16)). Our prediction (0.218 m/s) in this limit ($\alpha_G \rightarrow 0$) is close to these values. The problem lies in the region of high gas volume fraction. Due to the effect of swarms of bubbles, we expect V_d to go to zero when α_G is one. However, the opposite trend is obtained in Fig. 4-21 (b).

Figure 4-21: Typical behavior of calculated C_0 and V_d (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

Flores *et al.* (1998) studied the drift-flux model in oil-water flow and expressed the drift velocity as a function of oil volume fraction. An expression of the same form can also be used in the gas-liquid flow* (Gomez *et al.*, 2000):

$$V_d = V_\infty (1 - \alpha_G)^{0.5}. \quad (4-17)$$

* A similar form is also presented by Zuber & Findlay (1965). Their value for the power ranges from 0 to 3.

The interesting point here is the trend of V_d at high α_G . This expression also indicates that $V_d \rightarrow 0$ as $\alpha_G \rightarrow 1.0$. Our simple linear model does not reproduce this behavior in that limit. Further investigation of V_d in high volume fractions is therefore needed.

4.7.2 Further Investigation of V_d as $\alpha_G \rightarrow 1$. Using the same data set, instead of applying our procedure to the entire range of volume fractions, we now consider only the data at high α_G . The results for C_0 and V_d when $\alpha_G \in [0.95, 1.0]$ are shown in Fig. 4-22. C_0 is seen to decrease from 1.05 to 1.0, which is consistent with our previous observation. In this range of α_G , V_d now trends toward zero, in contrast to the previous increasing trend. The comparison between the calculated and experimental α_G displays a very high correlation. This shows that, although our previous linear model for the entire range of α_G provided good global accuracy, it failed to capture the correct behavior in limit $\alpha_G \rightarrow 1.0$.

Figure 4-22: Behavior of C_0 and V_d at high $\alpha_G \in [0.95, 1.0]$ (Data: Spedding & Nguyen (1976), air-water system, vertical flow, $D=1.79$ inch)

We now apply the same procedure to other inclination angles. In Fig. 4-23, similar behavior is observed for upward and horizontal flows. However, we cannot infer from this data where the inflection point in V_d should be --- for some data sets, we observe a decreasing trend only for $\alpha_G > 0.95$, while for other data sets, this behavior is observed for $\alpha_G > 0.8$.

Fig. 4-24 illustrates our results for downward flows. The trends are similar to those observed in upward flow, though negative drift velocities occur for downward flow. This is expected since drift velocity can be thought of as gas rise velocity.

Figure 4-23: Behavior of V_d at high α_G for upward and horizontal flows (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

Figure 4-24: Behavior of V_d at high α_G for downward flows (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

4.7.3 Effects of Inclination Angles on V_d . It is important to be able to predict the effect of pipe inclination angle on the drift velocity. We demonstrated this effect in the range of high gas volume fractions in the previous section. Hasan & Kabir (1999) developed a formula for the terminal gas rise velocity for oil-water flow, which is valid for upward flow for $\theta' \leq 70^\circ$ (θ' is measured from vertical):

$$V_{\infty}^{\theta'} = V_{\infty} \sqrt{\cos \theta' (1 + \sin \theta')^2}, \quad (4-18)$$

where V_{∞} is the terminal gas rise velocity in vertical upward flow. We introduce a simple extension of Hasan & Kabir's correlation for downward flow:

$$V_{\infty\theta'} = -V_{\infty} \sqrt{-\cos \theta' (1 + \sin \theta')^2}, \quad (4-19)$$

since a negative drift velocity is expected in downward flow.

In this work, we use the data of Spedding & Nguyen (1976) to compute the drift velocities in each inclination. The dependency of V_d on inclination is shown in Fig. 4-25. Note that Hasan & Kabir's correlation is for V_{∞} , that is V_d as α_G approaches zero. Our calculations use the data over

the whole range of α_G . However, our result displays a trend similar to that of Eq. (4-18). For downward flow, as discussed above, the fit between the predicted and experimental α_G is not very accurate so our parameters may be only approximate in this range.

A more accurate way of investigating the influence of inclination is to calculate V_d in several ranges of gas volume fraction, as shown in Fig. 4-26. In downward flow, the intermittent flow regime is replaced by stratified flow, so only high gas volume fractions are observed in downward flow. One observation we can make from Fig. 4-26 is that for upward flow, the drift velocity does not depend strongly on α_G .

Figure 4-25: V_d in different inclination angles (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

Figure 4-26: V_d in different ranges of α_G in different inclination angles (Data: Spedding & Nguyen (1976), air-water system, $D=1.79$ inch)

4.8 Evaluation of the Drift-Flux Model in Eclipse

The drift-flux model is used widely in reservoir simulators when the wellbore flow is coupled with the reservoir flow. One example is in the Multi-Segment Well calculations (Schlumberger GeoQuest, 2000) in Eclipse. Here, we use our data to evaluate the drift-flux model in Eclipse. In the Eclipse formulation, C_0 depends not only on α_G , but also on V_M . V_d is taken to be a function of α_G , V_M and θ . The detailed description for the development and formulation can be found in the Eclipse reference manual (Schlumberger GeoQuest, 2000). Recall that our development only considers the dependency of the parameters on α_G .

In Fig. 4-27, we plot the curve C_0 vs. α_G using the Eclipse model. The lines with different colors correspond to different mixture velocities V_M . In Fig. 4-27(a), we see the mixture velocity has a very strong effect on the value of C_0 . For example, when α_G is equal to 0.6, increasing V_M to 20 m/s makes C_0 equal to 1.0. Although the velocity and concentration profiles tend to distribute more uniformly at high flow rates, the high gas volume fraction appears to have a

stronger impact on the distribution profile. This is illustrated by the value of C_0 calculated from our data, shown as points in Fig. 4-27. For different ranges of α_G , a constant value for C_0 can be obtained, and the average α_G and average V_M can also be calculated. For the values for C_0 associated with the three highest V_M (19.7 m/s, 25.0 m/s and 38.6 m/s), we obtain C_0 greater than 1 except when α_G is extremely close to 1. It is demonstrated that $C_0 \rightarrow 1$ only when $\alpha_G \rightarrow 1$. This is in contrast to the Eclipse model, which shows a strong dependency on V_M that forces the C_0 curves to approach one quickly.

There are three user-definable parameters in the Eclipse model (A , B and F_v) that can be used to adjust the value of C_0 . The parameter A is the value of C_0 at low values of α_G and V_M . B is the value of the gas volume fraction at which C_0 will reduce from the value A . F_v adjusts the sensitivity of the C_0 curve to the mixture velocity. In Fig. 4-27(b), we tune these parameters to obtain a better match. Essentially, we modify the parameters to reduce the effect of V_M . The five curves corresponding to different mixture velocities now collapse to a single curve. This shows that the impact of the mixture velocity is overestimated for this data. However, it should be noted that this data is for pipes of diameter 1-2 inches. It is possible that V_M will have a greater effect on C_0 for larger diameter pipe.

The relationship between α_G and V_M is demonstrated in Fig. 4-28, which is obtained in a similar manner as Fig. 4-4 --- the average mixture velocity is computed in each flow pattern. We see that the high volume fraction corresponds to high mixture velocity. This relation is also displayed clearly in the flow pattern map in Fig. 4-7 --- the annular-mist flow cannot be expected to occur in low flow rates. Therefore, α_G can be expected to represent V_M , suggesting that further dependency on V_M is not required.

Fig. 4-29 displays the correlation for V_d used in Eclipse. The correction for θ dependency is achieved using Hasan & Kabir's formula (Eq. (4-18)). The behavior of V_d at the limiting values of α_G ($\alpha_G \rightarrow 0$ and $\alpha_G \rightarrow 1$) is consistent with our observations. The V_d curve in Eclipse provides an alternative when a more complicated model is required, rather than our simple linear model for V_d .

Figure 4-27: Comparison of C_0 between Eclipse correlation and calculated values for different V_M (Data: Spedding & Nguyen (1976), air-water system, vertical flow $D=1.79$ inch)

Figure 4-28: Relation between α_G and V_M (Data: Spedding & Nguyen (1976), air-water system, vertical flow $D=1.79$ inch)

Figure 4-29: V_d curve in Eclipse (air-water system, $D=1.79$ inch)

We now evaluate the correlation in Eclipse using our current data. The performance is shown in Fig. 4-30. Fig. 4-30 (a) displays the comparison using default values in Eclipse, while Fig. 4-30 (b) is that for the modified values suggested above. A noticeable improvement is

obtained at high gas volume fractions. This is due to the correction of C_0 at high α_G , as indicated in Fig. 4-27.

Figure 4-30: Performance of the drift-flux model correlation in Eclipse (Data: SU66, Govier *et al.* (1957), air-water system, vertical flow, $D=1.02$ inch)

4.9 Conclusions and Recommendations for Future Work

The drift-flux model is a simple but useful way of calculating the holdup. The use of appropriate values for C_0 and V_d determines the accuracy of this model. It was observed that this model may be applied to all flow patterns, and that flow patterns have a strong impact on the values for C_0 and V_d . A method was proposed to develop correlations for C_0 and V_d in which both C_0 and V_d are functions of α_G . This is reasonable since α_G was shown to approximately represent the flow pattern information. The two parameters were determined by directly minimizing the errors between the experimental and the estimated α_G . Compared to the previous approach, minimizing the objective function of the actual gas velocity, significant improvement was achieved. The performance of the drift-flux correlations in Eclipse was also evaluated using the current data. We showed that the effect of the mixture velocity is overestimated in the Eclipse model, and specific user-definable parameters were suggested to improve the prediction of α_G at high gas volume fraction.

The resulting C_0 , which has a value close to 1 at high α_G , is consistent with its physical meaning. However, our simple linear model for V_d does not account for the physical behavior of V_d as $\alpha_G \rightarrow 1$. A more realistic model is needed to capture behavior in this limit correctly.

The extension of drift-flux modeling is a promising research area. The drift-flux model needs to be further assessed for horizontal flow and downward flow, where the stratified flow pattern occurs. The method proposed in this work can be applied to a variety of other conditions. However, before it is generalized to wider conditions, the behavior of V_d as $\alpha_G \rightarrow 1$ should be further studied. The linear model should also be extended --- one way of doing this is to apply the procedure developed here in different ranges of α_G . The flow pattern information could be used to determine the appropriate ranges. The dependency of V_d on θ is another problem that deserves further attention. Improvement of the drift-flux model along the lines described here will ultimately result in more accurate well models in reservoir simulators.

Nomenclature

A	=	Pipe cross sectional area
C	=	Input volume fraction
C_0	=	Distribution parameter, profile parameter
D	=	Pipe internal diameter
g	=	Gravitational acceleration
p	=	Pressure
Q	=	Volumetric flow rate
Re	=	Reynolds number
V_d	=	Drift velocity of gas
V_G	=	Actual gas velocity
V_L	=	Actual liquid velocity
V_{SG}	=	Superficial gas velocity
V_{SL}	=	Superficial liquid velocity
V_M	=	Volumetric flux of the mixture
V_{∞}	=	Terminal gas rise velocity in vertical flow
$V_{\infty}^{\theta'}$	=	Terminal gas rise velocity in inclined pipe
x	=	Axial coordinate in pipe

Greek Letters

α	=	In situ volume fraction
θ	=	Pipe inclination angle (measured from horizontal)
θ'	=	Pipe inclination angle (measured from vertical)
μ	=	Dynamic fluid viscosity
ν	=	Kinematic fluid viscosity
ρ	=	Fluid density

Subscripts

G = Gas phase

L = Liquid phase

Acknowledgments

We are grateful to Dr. Jon Holmes (Schlumberger GeoQuest) for providing us with details on the drift-flux modeling procedure used in Eclipse.

4.10 References

- Barnea, D.: "A Unified Model for Predicting Flow-Pattern Transitions for the Whole Range of Pipe Inclinations," *Int. J. Multiphase Flow*, **13**, 1-12, 1987.
- Chen, J. J. J. and Spedding, P. L.: "Data on Holdup, Pressure Loss and Flow Pattern in a Horizontal Pipe," Report 214, University of Auckland, Auckland, New Zealand, 1979.
- Chen, X. T., Cai, X. D. and Brill, J. P.: "A General Model for Transition to Dispersed Bubble Flow," *Chem. Eng. Sci.*, **52**, 4373-4380, 1997.
- Cheremisinoff, N. P.: *Encyclopedia of Fluid Mechanics, Volume 3, Gas-Liquid Flows*, Gulf Publishing Company, Houston, Texas, 1986.
- Flores, J.G., Sarica, C., Chen, T.X. and Brill, J.P.: "Investigation of Holdup and Pressure Drop Behavior for Oil-Water Flow in Vertical and Deviated Wells," *J. Energy Resources Technology, Trans. ASME*, **120**, 8-14, 1998.
- Franca, F. and Lahey, R. T.: "The Use of Drift - Flux Techniques for the Analysis of Horizontal Two-Phase Flows," *Int. J. Multiphase Flow*, **18**, 887-801, 1992.
- Gomez, L. E., Shoham, O., Schmidt, Z., Chokshi, R. N. and Northug, T.: "Unified Mechanistic Model for Steady-State Two-Phase Flow: Horizontal to Vertical Upward Flow," *SPE J*, **5**, 339-350, 2000.
- Govier, G. W. and Aziz, K.: *The Flow of Complex Mixtures in Pipes*, Van Nostrand Reinhold Company, New York, NY, 1972.
- Govier G. W., Radford, B. A. and Dunn, S. C.: *Canadian J. Chem. Eng.*, **35**, 58, 1957.
- Hasan, A. and Kabir, C. S.: "A Simplified Model for Oil/Water Flow in Vertical and Deviated Wellbores," *SPE Production & Facilities*, **14**, 56-62, 1999.
- Kaya, A. S., Sarica, C. and Brill, J. P.: "Comprehensive Mechanistic Model of Two-Phase Flow in Deviated Wells," paper SPE 56522 presented at the SPE Annual Technical Conference and Exhibition, Houston, TX, October 3-6, 1999.
- Li, Q., Yi, D. and Wang, N.: *Modern Numerical Analysis* (in Chinese), Higher Education Press, Beijing, China, 1995.

- Mishima, K. and Ishii, M.: "Flow Regime Transition Criteria for Upward Two-Phase Flow in Vertical Tubes," *Int. J. Multiphase Flow*, **27**, 723-737, 1984.
- Mukherjee, H.: "An Experimental Study of Inclined Two Phase Flow," University of Tulsa, 1979.
- Ouyang, L-B.: "Single Phase and Multiphase Fluid Flow in Horizontal Wells," Ph.D. thesis, Stanford University, 1998.
- Petalas, N. and Aziz, K.: "A Mechanistic Model for Multiphase Flow in Pipes," CIM 98-39, Proceedings, 49th Annual Technical Meeting of the Petroleum Society of the CIM, Calgary, Canada, June 8-10, 1998.
- Petalas, N. and Aziz, K.: "A Mechanistic Model for Stabilized Multiphase Flow in Pipes," Stanford University, 1997.
- Schlumberger GeoQuest: "Eclipse 200 Reference Manual – Multi-Segment Wells," 2000.
- Spedding, P. L. and Nguyen, V. T.: "Data on Holdup, Pressure Loss and Flow Pattern for Two-Phase Air-Water Flow in an Inclined Pipe," Report 122, University of Auckland, Auckland, New Zealand, 1976.
- Zuber, N. and Findlay, J. A.: "Average Volumetric Concentration in Two-Phase Flow Systems," *J. Heat Transfer, Trans. ASME*, **87**, 453-468, 1965.

Part III. Novel Approaches to Account for Heterogeneities in the Vicinity of Non-Conventional Wells

Reservoir heterogeneity can have a significant influence on the performance of non-conventional wells. Detailed geocellular (or geostatistical) models are often built in an attempt to capture these important effects. However, the resulting models may be too detailed for full finite difference simulation, so alternative procedures or additional preprocessing steps must be applied.

In this part of the report, we present two methodologies for addressing this problem. The first procedure, described in Chapter 5, involves the use of a semi-analytical method for the modeling of non-conventional wells. The method is limited to single phase flow problems, though it does account for fine grid reservoir heterogeneity as well as the effects of wellbore hydraulics and downhole inflow control devices (e.g., downhole chokes). Such devices are an important component of “smart well” technology so it is important to develop models to predict their performance and quantify their potential benefits. Modeling of complex wells can also be accomplished via finite difference methods. Though usually more time consuming to build and run than semi-analytical models, finite difference procedures represent the most general approach for reservoir simulation. In Chapter 6 we present an approach for upscaling with irregular quadrilateral cells (i.e., general curvilinear, non-orthogonal grids). Such models result when general gridding techniques are applied to models with complex heterogeneity or complex geometric features such as NCW’s.

5 Semi-Analytical Modeling of Complex Wells with Downhole Inflow Control Devices

by Per H. Valvatne, Louis J. Durlofsky and Khalid Aziz

5.1 Introduction

The use of horizontal and other non-conventional wells has had a dramatic impact on the industry over the last decade. These wells greatly increase reservoir exposure and can increase the production from a single well significantly. However, non-conventional well performance can be impacted considerably by reservoir heterogeneity. Fluvial reservoirs, for example, typically display very large differences in permeability depending on whether the rock type is a channel sand or not, with the result that most of the inflow into a horizontal well might originate from a small

fraction of the perforated length. In layered or faulted reservoirs additional complications can arise; e.g., the formation permeability and pressure may vary so much that a single well completion might not be feasible.

These complications can be addressed through the use of complex well configurations coupled with so-called “intelligent completions.” A key aspect of this technology is the use of surface adjustable downhole chokes, which allow for the isolation or control of individual reservoirs or zones within a single reservoir. The downhole system makes use of packers that are set between the liner and tubing. It is then possible to control the inflow from different zones along the wellbore through the use of downhole chokes, across which a pressure drop can be imposed. This type of completion has been successfully installed and operated in several field applications (Lie and Wallace, 2000).

Downhole inflow control devices are very expensive, and it is important to be able to model and predict the performance of these complex systems. Finite difference reservoir simulators can be used to provide an in-depth analysis of the problem (Holmes et al., 1998), though there are several complications with this type of simulation. Specifically, finite difference models have large data requirements, especially when introducing complex well configurations. As a result, the tool might not be appropriate for quick screening studies. Further, because the reservoir is discretized, well connection factors (or well indexes), linking the well to the discrete model, are required. Default connection factors are typically based on the traditional Peaceman expression (Peaceman, 1978, 1991), developed under the assumption of two dimensional flow in a homogeneous reservoir. These well indexes may not be valid for complex wells with downhole chokes operating in heterogeneous reservoirs, particularly when the well is not aligned with the simulation grid.

The complications with the finite difference approach motivated us to investigate a simpler semi-analytical technique. Analytical and semi-analytical solutions to the pressure equation have a long history in the petroleum industry, especially within the field of transient well testing (Gringarten and Ramey, 1973). These solutions, expressed in terms of Green’s functions, can be combined using superposition to enable the modeling of complex well configurations (e.g., Gringarten and Ramey, 1973; Economides et al., 1996; Ouyang and Aziz, 1998; Maizeret, 1996; Ozkan et al., 1999; Tang et al., 2000). The well can then be modeled using a discrete number of well segments. Several implementations based on this approach have been presented (as dis-

cussed in the references indicated above). The effects of wellbore pressure losses on well performance are included in some of these methodologies (e.g., Ouyang and Aziz, 1998; Ozkan et al., 1999; Tang et al., 2000). In recent work (Wolfsteiner et al., 2000a, 2000b; Durlofsky, 2000), our group introduced an approximate method for representing reservoir heterogeneity, which we call the s - k^* permeability representation. This representation was implemented into the semi-analytical formulation and was shown to provide results in reasonable agreement with finite difference simulations for a variety of problems.

In this chapter, we extend existing semi-analytical approaches in several important directions. Most significantly, we show how downhole choke models can be coupled to the reservoir inflow equations. We implement a full Newton procedure to solve the resulting tightly coupled nonlinear system. In addition, we present a modified means for approximating the effects of heterogeneity. This new approach provides improved numerical stability over that of our previous approach (Wolfsteiner et al., 2000a, 2000b) and allows the s - k^* representation to be used for the highly heterogeneous fluvial reservoir descriptions considered here. Though the models currently used for pressure losses due to wellbore friction, acceleration, inflow, and downhole chokes are relatively simple, the general framework allows for the use of any model or correlation.

The solutions developed using our semi-analytical approach are restricted to simple reservoir geometries and single phase flow conditions. For more complex systems (e.g., two or three phase flow), finite difference simulation would be required. However, our method could be used to provide accurate well indexes to the finite difference simulator. See Palagi and Aziz (1992) (and references therein) for a discussion of how analytical (or semi-analytical) solutions can be used to generate well indexes for numerical simulation.

This chapter proceeds as follows. We first discuss the basic semi-analytical approach including our new treatment of reservoir heterogeneity. We then present wellbore flow and downhole choke models coupled with the reservoir flow equations and describe how the full nonlinear system is treated numerically. Next, the method is applied to several complex cases involving reservoir heterogeneity, downhole chokes and multiple reservoirs. The applicability of the method for the modeling of these complex scenarios is further established through comparisons to finite difference simulations when appropriate.

5.2 Modeling of Reservoir Inflow

We are concerned with the flow of a slightly compressible fluid in a porous medium. The pressure equation in this case is expressed as

$$k_x \frac{\partial^2 \Phi}{\partial x^2} + k_y \frac{\partial^2 \Phi}{\partial y^2} + k_z \frac{\partial^2 \Phi}{\partial z^2} = \phi \mu c_t \frac{\partial \Phi}{\partial t} , \quad (5-1)$$

where Φ , the potential, is given by $\Phi = p + (g/g_c)\rho h$ and $\mathbf{k}$ is the permeability tensor, assumed to be diagonal. Other variables are porosity ϕ , viscosity μ , density ρ and total compressibility c_t . To simplify the solution of the pressure equation the reservoir dimensions are transformed such that the reservoir becomes isotropic (Besson, 1990), with the scalar permeability k_g given by $k_g = (k_x k_y k_z)^{1/3}$.

The semi-analytical (Green's function) representation that underlies our overall approach has been described previously (Economides et al., 1996; Ouyang and Aziz, 1998; Maizeret, 1996; Ozkan et al., 1999; Tang et al., 2000; Wolfsteiner et al., 2000a, 2000b). Our description here will therefore be relatively brief and will emphasize new aspects of the formulation as well as some details required for our description of the residual and Jacobian construction below. Full details can be found in Valvatne (2000). Dimensionless inflow q_{ID} for a segment i_s , belonging to well i_w , is given by

$$q_{ID}(i_w, i_s) = \frac{q_I(i_w, i_s) B \mu}{C_q k_g x_e \Phi_i} , \quad (5-2)$$

where B is the formation volume factor, Φ_i is the initial reservoir potential and $q_I(i_w, i_s)$ is the inflow to segment i_s . The constant factor C_q is equal to 887.25 in field units. The expression is scaled by the reservoir extent in the x -direction x_e . Dimensionless potential Φ_D due to production/injection from segment i_s is then given by

$$\Phi_D(i_w, i_s) = \frac{\Phi_i - \Phi}{q_{ID}(i_w, i_s) \Phi_i} , \quad (5-3)$$

which can be expressed in terms of Green's functions as indicated above.

Using the Green's function representation of Φ_D , the dimensionless wellbore potential Φ_{wD} can be computed using the principle of superposition. The wellbore potential in segment (i_w, i_s) is given by a linear combination of the contributions from all segments in all wells:

$$q_{ID}(i_w, i_s) \Phi_{wD}(i_w, i_s) = \sum_{j_w=1}^{n_w} \sum_{j_s=1}^{n_s(j_w)} q_{ID}(j_w, j_s) \Phi_D(j_w, j_s, i_w, i_s). \quad (5-4)$$

The potential $\Phi_D(j_w, j_s, i_w, i_s)$ is the response at segment i_s , belonging to well i_w , due to production/injection from a segment (j_w, j_s) . In the absence of wellbore hydraulics the potential will be constant along the well. The point (i_w, i_s) is taken to be the segment midpoint, with a radial displacement away from the centerline equal to the wellbore radius r_w .

Our model for heterogeneity involves the use of an effective skin, introduced to capture the effects of near-well permeability fluctuations (Wolfsteiner et al., 2000a, 2000b; Durlofsky, 2000). Wolfsteiner et al. (2000a) described previously how skin is actually calculated; here we describe our new procedure for introducing this skin into the formulation.

Skin is introduced through the addition of a term of the form $q_{ID}(i_w, i_s)s(i_w, i_s)$ on the right hand side of Eq. 5-4, where $s(i_w, i_s)$ is the skin associated with well segment (i_w, i_s) . The underlying assumption in this approach is that the skin around segment (i_w, i_s) does not affect the contributions from other segments. In the resulting matrix, skin is therefore added only to the diagonal. This procedure, valid for positive or moderately negative skins, provided stable and accurate results in our previous calculations involving log-normally distributed geostatistical permeability fields characterized by spherical variograms.

Using this procedure, numerical problems can appear if skin is very negative in some segments of the well. This occurs, for example, in fluvial reservoirs that display a high degree of permeability variation over small distances. In these cases, the local skin can become highly negative and the potential contribution from neighboring segments can become dominant. This in turn creates instability in the solution. We found that a more stable approach is to use an effective wellbore radius r_{we} , rather than the actual radius r_w , where $r_{we} = r_w e^{-s}$. The effective wellbore radius for each segment will now vary depending on the magnitude of the local skin. In most cases, this method ensures that the matrix is diagonally dominant. For the most difficult cases, a rescaling of skin is necessary to ensure numerical stability (Valvatne, 2000). This rescaling reduces the magnitude of the most negative $s(i_w, i_s)$ to the value required for the stability of the numerical scheme.

As discussed by Wolfsteiner et al. (2000a), our approximate $s\text{-}\mathbf{k}^*$ representation of reservoir heterogeneity models the permeability field in terms of a constant, anisotropic background ef-

fective permeability k^* and a near-well effective skin s . The background permeability is simply the effective large-scale permeability for linear flow, while the skin captures the effect of the “altered” or near-well permeability on the well performance. For the fluvial systems considered here, we found the best accuracy when we took the altered zone to be quite small; i.e., approximately the channel width (Valvatne, 2000). Computing skin in this way and then introducing this effect into Eq. 5-4, as described above, provides an approximate but reasonably accurate means for representing permeability heterogeneity within the semi-analytical framework. See our previous papers for examples of the general level of accuracy of this approach for a variety of geostatistical systems.

5.3 Modeling of Downhole Pressure Losses

We now briefly describe the models incorporated into our technique for the modeling of pressure losses due to wellbore flow and downhole chokes. The models used here are quite simple, though we emphasize that other models or correlations can be introduced easily into the general formulation.

For single-phase flow in the wellbore, there will be frictional (Δp_f), acceleration (Δp_a) and hydrostatic pressure losses. Hydrostatic effects are included automatically when losses are expressed in terms of potential rather than pressure. Significant acceleration effects occur in segments receiving large inflows. For a well segment of length L , diameter D , axial flow rate q and inflow q_I , the losses are given by Ouyang et al. (1998):

$$\Delta p_f = \frac{C_f f L \rho q^2}{D^5} , \quad (5-5)$$

$$\Delta p_a = \frac{C_f \rho q q_I}{D^4} , \quad (5-6)$$

where f is the friction factor and the constant factor C_f is 2.956×10^{-12} in field units.

In this work we apply the formula suggested by Haaland (1981) to estimate f_0 , the friction factor for turbulent flow in the absence of inflow or outflow. This friction factor requires modification to account for the effects of inflow or outflow from the well segment. The corrections suggested by Ouyang et al. (1998), which represent the wellbore friction (for well inflow) in the form:

$$f = f_0 \left(1 - 0.0153 Re_w^{0.3978} \right) , \quad (5-7)$$

where Re_w is the wall Reynolds number (Reynolds number based on influx), were applied in this work.

Because downhole chokes are relatively new inventions, specific correlations for predicting pressure loss through these devices are not available in the literature. Additional complications arise because choking can be achieved using different methods depending on the system used. Sachdeva et al. (1986) proposed a two-phase choke model that is applicable for both critical and sub-critical flow (critical flow occurs when the flow rate through the choke is not affected by the upstream pressure). Because single-phase liquid flow at downhole conditions will rarely be critical, a simplified version of the model, applicable to single-phase, sub-critical liquid flow, was implemented. This model represents pressure losses through the choke Δp_c via (Schlumberger, 1999a):

$$\Delta p_c = \frac{C_c \rho q^2}{C_d^2 D_c^4} , \quad (5-8)$$

where D_c is the choke diameter and the discharge coefficient, C_d , is taken to be 0.85. The unit conversion constant C_c is equal to 1.532×10^{-8} in field units.

5.4 Residual and Jacobian Construction

We have now described the basic semi-analytical formulation, our representation of reservoir heterogeneity in terms of $s(i_w, i_s)$ and $\mathbf{k}^*$, and the models for pressure losses in the wellbore and downhole chokes. In the absence of downhole pressure losses, the governing system of equations is linear. However, inclusion of the effects of wellbore and downhole choke hydraulics renders the overall system nonlinear. In previous formulations, chokes were not included and the effects of wellbore hydraulics were generally secondary. Therefore, the coupling between the reservoir and the wellbore flow was relatively loose and a successive substitution type of iterative approach could be applied to solve the nonlinear system (Ouyang and Aziz, 1998; Wolfsteiner et al., 2000a, 2000b).

When downhole devices across which there can be very large pressure drops are included in the formulation, the coupling between the reservoir and the downhole hydraulics can be very tight. In such cases the successive substitution iterative approach may encounter difficulty con-

verging. Therefore, we formulate the problem here as a fully coupled nonlinear system and solve using Newton's method; i.e.,

$$\mathbf{J}\delta = -\mathbf{R} , \quad (5-9)$$

where $\mathbf{J}$ is the Jacobian matrix, $\mathbf{R}$ the residual vector and δ is the update to the solution vector. Elements of the Jacobian matrix are defined via $J_{ij} = \partial R_i / \partial x_j$, where $\mathbf{x}$ represents the vector of unknowns. The problem consists of potential residual equations R_p , equal to the total number of inflow segments n_{is} , and mass balance residual equations R_m , equal to the number of wells n_w . The unknowns are segment inflows q_I and wellbore potentials Φ_w .

The potential equation $R_p(i_w, i_s)$ for a well i_w and segment i_s is similar to Eq. 5-4, but with the added complexity that wellbore potential is no longer constant along the well, due to friction and acceleration. When multiple reservoirs are present in the problem, it is more convenient to use dimensional residual equations. The dimensional and expanded forms of the equations are then given by:

$$\begin{aligned} R_p(i_w, i_s) = & \sum_{j_w=1}^{n_w} \sum_{j_s=1}^{n_{is}(j_w)} \left(\frac{B\mu q_I(j_w, j_s)}{k^* x_e \Phi_i} \right) \Phi_D(j_w, j_s, i_w, i_s) \\ & - \frac{1}{\Phi_i} \left[\Phi_i - \Phi_w(i_w) - \sum_{k_s=1}^{k_s=i_s} (\Delta p_f(i_w, k_s) + \Delta p_a(i_w, k_s)) \right] = 0 \end{aligned} \quad (5-10)$$

$$R_m(i_w) = \sum_{i_s=1}^{n_{is}(i_w)} q_I(i_w, i_s) - q(i_w) = 0 . \quad (5-11)$$

Here $\Phi_w(i_w)$ is the wellbore potential for the well i_w , measured at the beginning of segment 1. It is convenient to keep $\Phi_D(j_w, j_s, i_w, i_s)$ in dimensionless form as this is computed directly from the dimensionless Green's function.

To form the Jacobian matrix, we require the derivatives of the residual equations with respect to the unknowns. For $j_w = i_w$ the derivatives are given by

$$\begin{aligned} \frac{\partial R_p(i_w, i_s)}{\partial q_I(j_w, j_s)} = & \left(\frac{B\mu}{k^* x_e \Phi_i} \right) \Phi_D(j_w, j_s, i_w, i_s) \\ & + \frac{1}{\Phi_i} \left[\sum_{k_s=1}^{k_s=i_s} \left(\frac{\partial \Delta p_f(i_w, k_s)}{\partial q_I(j_w, j_s)} + \frac{\partial \Delta p_a(i_w, k_s)}{\partial q_I(j_w, j_s)} \right) \right] \end{aligned} \quad (5-12)$$

$$\frac{\partial R_p(i_w, i_s)}{\partial \Phi_w(j_w)} = \frac{1}{\Phi_i} \quad (5-13)$$

$$\frac{\partial R_m(i_w)}{\partial q_l(j_w, j_s)} = 1 \quad (5-14)$$

$$\frac{\partial R_m(i_w)}{\partial \Phi_w(j_w)} = 0 \quad (5-15)$$

In the case when $j_w \neq i_w$, the expressions above will be simpler. Specifically, Eq. 5-12 no longer involves the wellbore friction and acceleration terms, and Eqs. 5-13 – 5-15 do not appear (i.e., the derivatives are all zero). See Valvatne (2000) for full details.

In the solution procedure, the Jacobian is recalculated at each iteration. This is computationally inexpensive, however, as the great majority of the execution time is spent evaluating the potentials $\Phi_D(j_w, j_s, i_w, i_s)$, given by the Green's functions. These are required only once for every time step and do not need to be updated when the Jacobian is recomputed. The inclusion of downhole devices like chokes will therefore have only a slight effect on the execution time, even if the problem becomes highly nonlinear.

The computation time required for the method scales with the number of well segments, in contrast to finite difference simulation, in which run time scales with the number of grid blocks. For the cases presented here, the method requires about 35 seconds, compared to about 10 minutes for finite difference simulation. We note also that the semi-analytical method does not require solutions at intermediate time steps (i.e., time-stepping is not performed), in contrast to finite difference simulation. This can provide additional savings relative to finite difference simulation.

The formulation as described up to this point is appropriate only when the total well rate (rather than wellbore pressure) is fixed in time. This limitation results from the form of the Green's function used in the formulation. Recently, a more general numerical treatment that allows the well rate to vary in time has been introduced (Serve, 2002). This will occur, for example, when the bottomhole pressure is fixed. This treatment is also necessary when the well penetrates multiple non-communicating reservoirs, in which case the flow rate from each reservoir can vary in time. This more general formulation entails the use of an additional superposition in time, introduced to account for the variation in time of the segment rates. With this new method-

ology, the system of equations can be formed to satisfy either a specified bottomhole pressure or a total rate constraint. The overall formulation is also appropriate for modeling steady state behavior, as might be approximated if a strong aquifer were present. In the examples below, we illustrate both steady state and transient calculations.

5.5 Application to Complex Reservoirs and Wells

In this section we briefly discuss the validation of the method and then apply it to a heterogeneous fluvial reservoir. We model a single reservoir, multiple reservoirs, and illustrate the effect of downhole inflow control devices.

5.5.1 Validation of the Overall Implementation. Validation of the specific components of the implementation (inflow, skin and wellbore hydraulics) was accomplished by comparing the solution, coded into a program called *Adwell*, to that obtained by the finite difference simulator *Eclipse* (Schlumberger, 1999b). The reservoir for these runs was homogeneous but anisotropic ($k_x=136$, $k_y=135$, $k_z=12$) and of dimensions $5,000 \times 6,5000 \times 300$ ft. The well was a 2,500 ft horizontal well, centered in the reservoir, with a radius of 0.15 ft. The well was modeled with 50 inflow segments in the semi-analytical model; the grid in the finite difference simulations contained $100 \times 130 \times 30$ blocks. Comparisons between the semi-analytical and finite difference results, for cases involving specified skins along the well and wellbore hydraulics, were excellent (see Valvatne et al., 2001, for detailed comparisons). We note that finite difference results in this case can be considered to be reliable, since the reservoir was homogeneous, the grid was sufficiently fine and the well was aligned with the simulation grid.

5.5.2 Single Heterogeneous Reservoir. The reservoir in this case, shown in Fig. 5-1, is fluvial-based and highly heterogeneous (Mao and Journel, 1999). The permeability distribution was generated using geostatistical techniques, with the channels introduced through use of object-based simulation (Deutsch, 1997). The overall mean permeability is 176 mD, though two distinct populations of permeability are clearly evident. The channels contain sandstone with a mean of about 200 mD, but with a long upper tail. The mudstone-dominated background has an average permeability of about 6 mD, creating a very sharp contrast between the background and channel

permeabilities. The components of the effective permeability tensor $\mathbf{k}^*$, obtained using a flow-based upscaling technique, are the same as those used for the homogeneous verification cases described above ($k_x^* = 136$, $k_y^* = 135$, $k_z^* = 12$). Average reservoir porosity is 0.17. The well and reservoir dimensions are also as described above. Initial reservoir potential is 3000 psi. Total system compressibility is $3 \times 10^{-6} \text{ psi}^{-1}$. The oil is of density $60 \text{ lb}_m/\text{ft}^3$. The reservoir boundaries are sealed (no flow).

The well location is indicated in Fig. 5-1. The central section of the well is completed in the channel, as is also evident from Fig. 5-2 where we plot the altered zone permeability (i.e., the permeability used for our skin calculation) along the well. The permeability contrast is very large along the well, varying from more than 500 mD to about 7 mD over a few feet. Inflow profiles computed using Eclipse and Adwell are shown in Fig. 5-3. The Eclipse model contained $100 \times 130 \times 30$ blocks. Again, because the well is aligned with the simulation grid, the simulation results are considered to be reliable in this case. Given the complexity of the permeability field and the approximate nature of our $s\text{-}\mathbf{k}^*$ representation of permeability heterogeneity, the agreement in inflow profiles between the two results is quite satisfactory. The bottomhole pressures for finite difference and semi-analytical calculations at a time of 100 days are 2,251 and 2,207 psi, corresponding to an error in predicted drawdown of less than 10% (average reservoir pressure at 100 days is 2,711 psi). We note that our new treatment of heterogeneity in the semi-analytical solution procedure (described above), in which we use the effective wellbore radius ($r_w e^{-s}$) in our calculations, is essential for this problem. Introducing the skin without this treatment results in solution instability in this case.

Figure 5-1: Fluvial reservoir permeability field and horizontal well

Figure 5-2: Altered permeability k_a in the near-well area

Figure 5-3: Inflow profiles in heterogeneous reservoir

Figure 5-4: Segment structure for modeling chokes

The high flow rates from high permeability channels in this reservoir would likely result in early water or gas breakthrough. From a reservoir management standpoint, one may want to reduce the production from the channel and obtain a more uniform influx. We now introduce downhole chokes into the model and show how they can be used to improve the inflow profile.

The well completion is modeled with separate segments representing tubing, annulus and chokes, as depicted in Fig. 5-4. For the case under study, it seems appropriate to use three inflow zones along the well, with the objective being to reduce inflow from the central section completed in the channel. The settings of the chokes are obtained by trial and error until an “optimized” inflow profile is achieved. Only the central inflow zone required choking. The optimized inflow profile, along with the wellbore pressure (at sand face) needed to achieve it, are shown in Figs. 5-5 and 5-6. From Fig. 5-6 and Table 5-1 we see that, by imposing a 709 psi pressure drop across the choke, it is possible to reduce the inflow from the channel from 22,748 to 11,622 stb/day. This represents a reduction from 75% to 39% of the total inflow. The inflow from zone 3 (see Fig. 5-5 for the exact location of the zones) is still low, representing only 17% of the total. The permeability in this section, which is less than 10 mD, is just too low for the inflow to increase to that of the other zones. Nonetheless, there is a clear flattening of the inflow profile along the well, which would be expected to result in a delay of water or gas breakthrough. The bottomhole pressure is, however, less when the well is choked, indicating that the well productivity index will be smaller for this case.

The convergence of the solution is very rapid in this case and was achieved after only five iterations. The solution is in relatively close agreement with Eclipse (Holmes et al. 1998; Schlumberger, 1999a, 1999b), as shown in Fig. 5-7. We note that the same segment structure was used in Eclipse as in Adwell, with choke pressure drop given through look-up tables.

Table 5-1: Optimized Inflow Using Downhole Chokes

	Inflow without choking (stb/day)	Inflow with choking (stb/day)
Zone 1	5,326	13,411
Zone 2	22,748	11,622
Zone 3	1,926	4,967

Figure 5-5: Inflow profiles with and without downhole choking

Figure 5-6: Wellbore pressure (at sand face) with and without choking

Figure 5-7: Comparison with Eclipse with downhole choking

5.5.3 Multiple Reservoirs. Formation properties such as permeability or pressure may vary considerably if the field is comprised of multiple reservoirs or fault blocks. Using downhole chokes it is possible to achieve commingled production since the different inflow zones do not need to be produced at the same wellbore pressure. This has in fact been the most common reason for installing downhole chokes (Lie and Wallace, 2000).

To demonstrate the application of our method to such cases, we consider a reservoir containing three non-communicating fault blocks (Fig. 5-8), with each fault having a vertical throw of 100 ft. The boundary conditions in the y -direction (front and back of the model) are constant pressure; in the other directions the boundaries are no-flow. This results in the reservoir achieving steady state conditions, as might be approximated if a strong aquifer were present. The reservoir dimensions, permeability field and fluid properties are the same as those used previously.

There is a 500 psi pressure differential across each fault, with the rightmost block having the highest initial potential of 3,500 psi. The well is horizontal and is completed over 1,000 ft in each block. The permeability along the well is again highly variable (Fig. 5-9). Conventional completions (i.e., completions without chokes) would not be feasible in this case due to the large pressure differences between fault blocks.

Results for the inflow profiles and wellbore pressures (at sand face), with and without choking, are shown in Figs. 5-10 and 5-11. As is evident from the figures, without choking the low pressure in the left block results in cross flow from the other blocks (apparent from the negative inflow along the first 1,000 ft of the well in Fig. 5-10). Using downhole chokes, it is possible to produce each reservoir with an optimal wellbore pressure, as shown in Figs. 5-10 and 5-11, such that the inflow is more uniform and is distributed according to the perforated well length (Table 5-2). The productivity index is again less when the well is choked.

In Fig. 5-12 we compare the optimized inflow profile computed using our model with that obtained using Eclipse. The agreement between the two results is again quite close. In some cases, such as when the well is not oriented along a coordinate direction or when the well is a complex multilateral, the results from our method might be expected to be more accurate than the results from Eclipse. This is because the finite difference results can lose accuracy when the well is skew to the grid or when the various branches of the multilateral interact strongly. The semi-analytical calculations, by contrast, do not suffer any additional inaccuracies in this case as they are not based on an underlying grid.

Figure 5-8: Reservoir geometry with fault blocks

Figure 5-9: Altered permeability k_a in the near-well area

Figure 5-10: Inflow profiles with and without choking in faulted reservoir

Figure 5-11: Wellbore pressure (at sand face) with and without choking in faulted reservoir

Table 5-2: Optimized Inflow in Faulted Reservoir

	Inflow without choking (stb/day)	Inflow with choking (stb/day)
Zone 1	-3,657	9,836
Zone 2	5,915	9,722
Zone 3	27,742	10,442

Figure 5-12: Comparison with Eclipse with downhole choking in faulted reservoir

5.5.4 Dual-lateral Well. Our last example involves a dual-lateral well, with each lateral penetrating a different channel in the reservoir. The reservoir model is the same as was used in the previous examples though the well is in a different layer in this case. The reservoir (in the plane of the well) and the well are shown in Fig. 5-13. Note that only the lateral branches of the well are perforated. The lower lateral (lateral 1) is about 1900 ft in length and is perforated along its entire length. The upper lateral (lateral 2) is of length 700 ft and is perforated only in the channel. Two chokes are also included in this model, located at the edges of the channels as indicated in the figure via circles. The well is specified to produce at a total production rate of 10,000 stb/d.

Our computed inflow profiles at a time of 100 days, with and without choking, are shown in Fig. 5-14. In the figure, we plot production rate per unit well length. When the well is not choked, lateral 1 produces an average of 2.46 stb/d/ft while lateral 2 produces an average of 7.54 stb/d/ft. Chokes are then set in an attempt to achieve a more nearly uniform (in terms of production rate per unit length) inflow profile. The inflow profile in this case is much flatter than in the case with no choking. Both laterals now produce an average of about 3.84 stb/d/ft. Thus, the objective of more uniform production is achieved through the use of downhole chokes.

Bottomhole pressure as a function of time for the two cases is shown in Fig. 5-15. It is apparent that there is a significant difference in bottomhole pressure, with the choked well displaying pressures about 570 psi less than those of the unchoked well. As was the case in the previous examples, choking has an adverse impact on the well productivity index. However, the more uniform inflow profile may more than compensate for this effect.

Figure 5-13: Fluvial reservoir permeability field and dual-lateral well

Figure 5-14: Inflow profiles with and without choking for dual-lateral well

Figure 5-15: Bottomhole pressure with and without choking for dual-lateral well

5.6 Conclusions

In this work, we addressed the detailed modeling of wells with downhole chokes. Rather than apply finite difference techniques, we extended the capabilities of semi-analytical methods based on Green's functions. Our approach is appropriate for single phase flow and includes the effects of downhole chokes and frictional and acceleration pressure losses in the wellbore. The effects of reservoir heterogeneity are approximated using an s - k^* permeability representation.

Because the problem becomes significantly nonlinear when downhole chokes are included in the formulation, we applied Newton's method to determine the inflow and pressure profiles for the fully coupled system. The accuracy of the overall formulation was demonstrated by comparing the results to detailed finite difference simulations. Reasonably close agreement between the semi-analytical results and finite difference results was obtained in all cases. In this work we used a highly heterogeneous fluvial reservoir model. This required us to modify the s - k^* procedure to ensure numerical stability.

Using several realistic well configurations we showed how downhole chokes can be a useful tool for improving reservoir performance. In the case of a horizontal well partially completed in a channel, the inflow from the channel was reduced from 75% to 39% of the total, achieving a more uniform inflow profile. Production from a faulted reservoir was optimized such that the inflow from each fault block reflected the perforated well length. Production via a dual-lateral well was similarly optimized. In the absence of downhole inflow control devices, such cases might require the use of multiple wells.

The results obtained using our approximate model of heterogeneity compare well with those obtained using the more complex finite difference simulation with detailed permeability fields. Given the simplicity of model creation, the quick solution time and the intrinsic high uncertainty of the permeability field, the results of our technique are clearly acceptable for a variety of applications. The method could be used both as a standalone application, to quickly predict the productivity from a wide range of well configurations and permeability realizations, or integrated with other well planning tools where a full finite difference simulator would be too complex. The ability of the method to fully account for effects from other wells and boundaries, combined with the fact that it is decoupled from any underlying grid, makes it an excellent candidate for the calculation of well indexes for use in finite difference simulators.

Nomenclature

B	formation volume factor, rb/stb
C	unit system conversion factor
c_t	total compressibility, 1/psi
D	diameter, ft
f	Fanning friction factor
g	acceleration due to gravity, lb _f .ft/sec ²
g_c	gravitational conversion constant, 32.2 lb _m /lb _f .ft/sec ²
h	height from datum depth, ft
J	Jacobian matrix
k	reservoir permeability, mD
k^*	effective reservoir permeability, mD
L	length along wellbore, ft
p	pressure, psi
q	flowrate, stb/day
R	residual equation
r_w	wellbore radius, ft
s	skin
t	time, days
x_e, y_e, z_e	reservoir dimensions, ft
δ	update vector
Φ	potential, psi
ϕ	porosity
μ	fluid viscosity, cp
ρ	fluid density, lb _m /ft ³

Subscripts

a	acceleration or altered
c	choke
D	dimensionless
d	discharge
e	effective or extent
f	friction
g	geometric average
I	inflow
i	initial
m	mass balance
p	potential
q	flow rate
w	well
x, y, z	coordinate direction

Acknowledgments

We are grateful to Jerome Serve (Stanford University) for generating the results for the dual-lateral case.

5.7 References

- Besson, J.: "Performance of Slanted and Horizontal Wells on an Anisotropic Medium," paper SPE 20965 presented at EUROPEC, Netherlands, Oct. 22-24, 1990.
- Deutsch, C.V.: "Developing Hierarchical Object-Based Stochastic Modeling of Fluvial Reservoirs," Stanford Center for Reservoir Forecasting Report, May, 1997.
- Durlofsky, L.J.: "An Approximate Model for Well Productivity in Heterogeneous Porous Media," *Math. Geol.*, **32**, 421-438, 2000.
- Economides, M.J., Brand, C.W. and Frick, T.P.: "Well Configurations in Anisotropic Reservoirs," *SPEFE*, 257-262, Dec. 1996.
- Gringarten, A.C. and Ramey, H.J.: "The Use of Source and Green's Functions in Solving Unsteady-Flow Problems in Reservoirs," *SPEJ*, 285-296, Oct. 1973.
- Haaland, S.E.: "Simple and Explicit Formula for the Friction Factor in Turbulent Pipe Flow Including Natural Gas Pipelines IFAG B-131," Technical Report, Division of Aero- and Gas Dynamics, The Norwegian Institute of Technology, Norway, 1981.
- Holmes, J.A., Barkve, T. and Lund, O.: "Application of a Multisegment Well Model to Simulate Flow in Advanced Wells," paper SPE 50646 presented at the SPE European Petroleum Conference, The Hague, Oct. 20-22, 1998.
- Lie, O.H. and Wallace W.: "Intelligent Recompletion Eliminates the Need for Additional Well," paper IADC/SPE 59210 presented at the IADC/SPE Drilling Conference, New Orleans, Feb. 23-25, 2000.
- Maizeret, P.D.: "Well Indices for Non-Conventional Wells," Master's Report, Stanford University, 1996.
- Mao, S. and Journel, A.G.: "Generation of a Reference Petrophysical / Seismic Data Set: The Stanford IV Reservoir," Stanford Center for Reservoir Forecasting Report, May, 1999.
- Ouyang, L.B., Arbabi, S. and Aziz, K.: "A Single-Phase Wellbore-Flow Model for Horizontal, Vertical, and Slanted Wells," *SPEJ*, 124-133, June, 1998.
- Ouyang, L.B. and Aziz, K.: "A Simplified Approach to Couple Wellbore Flow and Reservoir Inflow for Arbitrary Well Configurations," paper SPE 48936 presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Sept. 27-30, 1998.
- Ozkan, E., Sarica, C. and Haci, M.: "Influence of Pressure Drop Along the Wellbore on Horizontal-Well Productivity," *SPEJ*, 288-301, Sept. 1999.
- Palagi, C.L. and Aziz, K.: "The Modeling of Vertical and Horizontal Wells with Voronoi Grid," paper SPE 24072 presented at the SPE Western Regional Meeting, Bakersfield, Mar. 30 – Apr. 1, 1992.

Peaceman, D.W.: "Interpretation of Well-Block Pressures in Numerical Reservoir Simulation," *SPEJ*, 183-194, June, 1978.

Peaceman, D.W.: "Representation of a Horizontal Well in Numerical Reservoir Simulation," paper SPE 21217 presented at the SPE Symposium on Reservoir Simulation, Anaheim, Feb. 17-20, 1991.

Sachdeva, R., Schmidt, Z., Brill, J.P. and Blais, R.M.: "Two-Phase Flow Through Chokes," paper SPE 15657 presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Oct. 5-8, 1986.

Schlumberger GeoQuest: "Eclipse 200 Reference Manual – Multi-Segment Wells," 99A Release, 1999a.

Schlumberger Geoquest: Eclipse Reservoir Simulator, 99A Release, 1999b.

Serve, J.: "Enhanced Semi-analytical Modeling of Complex Well Configurations," Master's Report, Stanford University, 2002 (in preparation).

Tang, Y., Ozkan, E., Kelkar, M., Sarica, C. and Yildiz, T.: "Performance of Horizontal Wells Completed with Slotted Liners and Perforations," paper SPE/PS-CIM 65516 presented at the SPE/Petroleum Society of CIM International Conference on Horizontal Well Technology, Calgary, Nov. 6-8, 2000.

Valvatne, P.H.: "A Framework for Modeling Complex Well Configurations," Master's Report, Stanford University, 2000.

Valvatne, P.H., Durlofsky, L.J. and Aziz, K.: "Semi-analytical Modeling of the Performance of Intelligent Well Completions," paper SPE 66368 presented at the SPE Reservoir Simulation Symposium, Houston, Feb. 11-14, 2001.

Wolfsteiner, C., Durlofsky, L.J. and Aziz, K.: "Approximate Model for Productivity of Nonconventional Wells in Heterogeneous Reservoirs," *SPEJ*, 218-226, June, 2000a.

Wolfsteiner, C., Durlofsky, L.J. and Aziz, K.: "Efficient Estimation of the Effects of Wellbore Hydraulics and Reservoir Heterogeneity on the Productivity of Non-Conventional Wells," paper SPE 59399 presented at the SPE Asia Pacific Conference on Integrated Modeling for Asset Management, Yokohama, Apr. 25-26, 2000b.

6 Numerical Calculation of Equivalent Cell Permeability Tensors for General Quadrilateral Control Volumes

by Chuanping He, Michael G. Edwards and Louis J. Durlofsky

6.1 Introduction

Geologic models of oil and gas reservoirs typically contain a much higher level of detail than can be accommodated in flow models. As a result, these geologic models must be coarsened, or up-scaled, for use in the flow simulator. The most fundamental quantity computed in the upscaling step is the coarse scale equivalent permeability tensor. This permeability is computed from the underlying fine scale permeabilities in such a way that the average (local) flow response computed using the coarse grid permeability replicates that obtained using the fine scale permeability description.

In the past, reservoir simulation models generally made use of gridding procedures in which the coarse scale simulation grid exactly overlaid the fine grid; i.e., all of the vertices of the coarse grid cells corresponded to fine grid cell vertices. The coarse grid is generated, in these cases, by simply removing grid lines from the fine grid system (Durlofsky et al., 1997). Recent flow based grid generation schemes, discussed in more detail below, produce coarse grids in which the grid lines do not correspond to the fine grid. These gridding approaches are illustrated in Fig. 6-1, which shows both types of grids (in the figures, finer lines correspond to the fine grid and heavier lines to the coarse grid).

The purpose of this paper is to develop and apply a method for the calculation of equivalent cell permeability tensors appropriate for the general two dimensional control volumes illustrated in Fig. 6-1b. By general control volumes, we mean non-rectangular quadrilateral and other polygonal cells. Although many methods for the calculation of equivalent grid block permeability tensors have been developed, few of these methods are directly applicable to the general control volumes considered here. Complications arising in the general case include geometric effects (e.g., coarse cells are non-rectangular) and the lack of an exact correspondence between fine and coarse grid cell boundaries. Both of these issues will be addressed in detail in the procedures developed here.

(a)

(b)

Figure 6-1: Fine and coarsened grids with (a) rectangular coarse blocks and (b) irregular coarse cells that do not overlay the fine grid

A variety of both numerical and analytical techniques for the calculation of equivalent grid block permeability tensors (designated $\mathbf{k}^*$) have been developed. Numerical approaches entail the solution of the local pressure equation over the fine scale region to be upscaled into a coarse grid cell. For thorough discussions of existing methodologies, see the reviews by Wen and Gomez-Hernandez (1996) and Renard and de Marsily (1997). For reservoir simulation applications, the more accurate numerical approaches are generally preferable to analytical approaches. This is because the calculation of $\mathbf{k}^*$ usually represents a very small cost relative to the overall simulation, so the incremental computational expense of using a numerical rather than a less accurate analytical approach is generally not a concern.

The numerical procedure developed in this work represents a significant generalization of previous techniques. The method entails the mixed finite element solution of the local fine scale pressure equation over the general quadrilateral or polygonal region corresponding to the coarse scale control volume. We triangulate the region (using the algorithm of Barber et al., 1996) to ensure that the cell boundaries are accurately resolved and that all internal fine grid permeability discontinuities are captured. Periodic boundary conditions are then applied (Durlofsky, 1991). This requires the determination of correspondences between element edges on opposite sides of the coarse grid quadrilateral, which we obtain by mapping to transform space. The resulting permeability tensor is computed via integration of the fine scale velocity and pressure gradient over the coarse cell volume. We refer to the overall upscaling method as the conforming scale up method because the technique ensures that the region over which $\mathbf{k}^*$ is computed conforms exactly to the coarse cell.

Other recent upscaling techniques that are based on finite element solutions include the formulations described by Zijl and Trykozko (2001) and Trykozko et al. (2000). These approaches, developed for both two and three dimensional systems, apply conventional nodal and mixed-hybrid finite element methods to obtain upper and lower bounds for the cell $\mathbf{k}^*$. Refinement of both solutions is used to provide converged results. This approach can be very useful when the coarse scale cell possesses a highly discontinuous subgrid permeability distribution. Zijl and Trykozko (2001) also describe different approaches for computing upscaled permeabilities (pressure-flux averaging, pressure-dissipation averaging or flux-dissipation averaging) and discuss their relative advantages and disadvantages.

Eek-Jensen et al. (1999) developed a scale up procedure for general quadrilaterals based on the use of a flux-continuous finite difference stencil. Edwards (1996) developed a related up-scaling approach for general geometry permeability tensors in which the coarse quadrilateral cell overlays the underlying fine cells. Though these and the methods of Zijl and Trykozko (2001) and Trykozko et al. (2000) address important aspects of the general upscaling problem, none of them considers the case where the coarse grid does not overlay the fine grid. This is a primary motivation for the conforming scale up method developed here.

As indicated above, the method we develop is appropriate for use with flow-based gridding schemes that generate simulation grids that do not overlay the fine scale geological grid. Within a reservoir simulation context, methods of this type have been developed by Tran (1995), Wen (1996), Verma and Aziz (1997), Edwards et al. (1998) and Castellini et al. (2000), among others (the work of Castellini et al., 2000 is described in Chapter 3 of this report). Though these methods differ in important respects, they all apply some type of flow information (e.g., streamlines and isopotentials) for the determination of the coarse grid structure. In none of these procedures, however, are the coarse scale cell permeabilities computed using a method as general as that developed here.

This chapter proceeds as follows. We first present the governing equations and numerical procedures for the calculation of k^* . Next, we apply the method to calculate k^* for coarse grid cells and then solve several large scale flow problems. We couple the method to a flow-based gridding procedure and generate global flow results using the combined methodology. Improved results, relative to those generated using simpler gridding and upscaling procedures, are obtained. We conclude with further discussion and suggestions for future research directions.

6.2 Governing Equations and Numerical Procedures

In this section we present the equations governing single phase flow in porous media, describe the mixed finite element method applied in this work, and discuss our treatment of geometric effects. We note that, although the procedures developed here entail the calculation of the single phase equivalent cell permeability k^* , they can be expected to be applicable to more general multiphase displacements. This is because of the close similarity between the single phase and multiphase pressure equations. Further details on some of the solution procedures can be found in He (2000).

6.2.1 Pressure Equation and Periodic Boundary Conditions. We consider steady, single phase incompressible flow. The governing equations, in dimensionless form, are the mass conservation equation and Darcy's law:

$$\nabla \cdot \mathbf{u} = 0 , \quad (6-1)$$

$$\mathbf{u} = -\mathbf{k}(\mathbf{x})\nabla p , \quad (6-2)$$

where $\mathbf{u}$ is the Darcy velocity, $\mathbf{k}(\mathbf{x})$ is the spatially varying permeability tensor ($\mathbf{x}$ designates spatial location) and p is pressure. Combining these equations gives the single phase pressure equation:

$$\nabla \cdot (\mathbf{k}(\mathbf{x})\nabla p) = 0 . \quad (6-3)$$

Our procedure for the calculation of $\mathbf{k}^*$ entails the solution of Eq. (6-3) over the fine scale region corresponding to a coarse scale cell. Prior to the solution of Eq. (6-3), boundary conditions must be specified. In this work, we apply periodic boundary conditions with a jump in pressure. Though many different types of boundary conditions have been proposed for the calculation of $\mathbf{k}^*$, periodic boundary conditions are considered to be among the more robust and accurate (see e.g., Renard and de Marsily, 1997; Durlofsky, 1991; Zijl and Trykozko, 2001; Boe, 1994; Pickup et al., 1994). We first present periodic boundary conditions for the case of rectangular coarse cells (for which their application is relatively straightforward) and then describe their application for a general control volume.

We specify an arbitrary large scale pressure gradient $\mathbf{G} = G_x \mathbf{i}_x + G_y \mathbf{i}_y$. To compute $\mathbf{k}^*$, we solve the local problem twice: once with $G_x \neq 0$ and $G_y = 0$, and once with $G_x = 0$ and $G_y \neq 0$. For the case with $G_x \neq 0$, the periodic boundary conditions for a square grid block with sides of unit length are as follows (with reference to Fig. 6-2):

$$p(x,0) = p(x,1) \quad \text{on } \partial D_1 \text{ and } \partial D_2 , \quad (6-4a)$$

$$p(0,y) = p(1,y) - G_x \quad \text{on } \partial D_3 \text{ and } \partial D_4 , \quad (6-4b)$$

$$\mathbf{u}(x,0) \cdot \mathbf{n}_1 = -\mathbf{u}(x,1) \cdot \mathbf{n}_2 \quad \text{on } \partial D_1 \text{ and } \partial D_2 , \quad (6-4c)$$

$$\mathbf{u}(0,y) \cdot \mathbf{n}_3 = -\mathbf{u}(1,y) \cdot \mathbf{n}_4 \quad \text{on } \partial D_3 \text{ and } \partial D_4 . \quad (6-4d)$$

The quantity $\mathbf{k}^*$ is then computed using an averaged form of Darcy's law,

$$\langle \mathbf{u} \rangle = -\mathbf{k}^* \cdot \mathbf{G}, \quad (6-5)$$

where $\langle \mathbf{u} \rangle$ is the average velocity through the unit cell, determined by integrating the fluxes over the cell sides. Note that the periodicity conditions guarantee that $\langle u_x \rangle$ is the same whether it is computed over side 3 or 4 (and similarly for $\langle u_y \rangle$). From Eq. (6-5), and the solutions of Eq. (6-3) subject to Eq. (6-4), we can determine the components of $\mathbf{k}^*$ unambiguously. See Durlofsky (1991) for further details.

Figure 6-2: Schematic of unit square solution domain

6.2.2 Finite Element Solution Procedure. We solve Eq. (6-3) subject to (6-4) using a lowest order mixed finite element method applied to a triangulated domain. Mixed methods are well suited to problems of this type because they provide accurate fluxes (as needed in the calculation of $\mathbf{k}^*$) even with highly discontinuous coefficients (i.e., large variations in $\mathbf{k}(\mathbf{x})$). Marini (1985) showed that the lowest order mixed method is identical to the lowest order non-conforming finite element method (NCFEM) for the solution of equations of the form of (6-3). We therefore apply the lowest order non-conforming finite element technique, which is simpler to implement than the mixed method, in this study. We note that the mixed-hybrid method, as used by Zijl and Trykozko (2001) and Trykozko et al. (2000), would provide improved computational efficiency. However, because the local problems solved here are relatively small, this is not a major concern in this work.

In the lowest order NCFEM, we represent the pressure, designated $\hat{p}$, in terms of unknown coefficients p_i and piecewise linear basis functions $\phi^i(x, y)$. The ϕ^i are defined within an element such that they are 1 at the midpoint of edge i and 0 at the other two edge midpoints in the triangular element. The pressure approximation is given via:

$$\hat{p} = \sum_{i=1}^N p_i \phi^i(x, y), \quad (6-6)$$

where N is the total number of element edges. By expressing the pressure equation in terms of p_i and ϕ^i and weighting with ϕ^k , we obtain the weak form of the equation:

$$\sum_{i=1}^N p_i \int_D (\mathbf{k} \cdot \nabla \phi^i) \cdot \nabla \phi^k dA + \int_{\partial D} (\mathbf{u} \cdot \mathbf{n}) \phi^k dl = 0. \quad (6-7)$$

Equation (6-7) describes a linear system of equations that can be solved to obtain the p_i . The approximation for the Darcy velocity at the edge midpoint, designated $\hat{\mathbf{u}}_i$, can then be computed via:

$$\hat{\mathbf{u}}_i = - \sum_{i=1}^N p_i (\mathbf{k} \cdot \nabla \phi^i). \quad (6-8)$$

By construction, flux is continuous over element edges in the non-conforming method. As a consequence, the implementation of periodic flux boundary conditions is simplified because the flux

can be calculated directly. A further aspect of the flux continuity condition is that the $\hat{\mathbf{u}}_i$ computed from Eq. (6-8) conserve mass over the element; i.e.,

$$\sum_{i=1}^3 \hat{\mathbf{u}}_i \cdot \mathbf{n}_i l_i = 0 ,$$

where $i=1,3$ designates the three edges in the element, l_i is the length of side i and $\mathbf{n}_i$ is the unit vector normal to side i .

6.2.3 Treatment of Geometric Effects: Quadrilateral Cells. Up to this point we have described the general upscaling procedure for rectangular coarse grid cells. We now consider the case of a general quadrilateral grid block. The case of a polygonal coarse scale control volume will be discussed briefly below. We assume the underlying fine scale permeability field is defined on a Cartesian grid. This assumption is introduced for convenience and is not a limitation of the method. The set of nodes that are used to define the grid for upscaling are comprised of the fine Cartesian grid nodes belonging to the coarse cell together with boundary nodes defined by the intersection of the Cartesian grid with the coarse cell boundaries. The local fine grid is then formed with respect to the coarse quadrilateral cell via a Delaunay triangulation, as illustrated in Fig. 6-3. The Delaunay triangulation, accomplished using `qhull` (Barber et al., 1996) is performed such that all internal permeability discontinuities and the coarse cell boundaries are preserved. Points that are too close together, as determined according to a prescribed tolerance, are merged before triangulation.

This triangulation defines the “fine” grid corresponding to the coarse cell. Fine grid permeabilities are then inserted directly from the underlying Cartesian cells into the corresponding grid of triangular elements. Note that, in the description below, we use “edge” to mean the edge of a triangular element (used to discretize the fine grid region corresponding to a coarse cell) and “side” to mean the side of the coarse scale control volume itself.

For irregular quadrilaterals with equal spacings of nodes on each side, pressure and flux correspondences of the form of Eq. (6-4) can be established. The precise relationships presented in Eq. (6-4) will require some modification to account for the non-rectangular geometry, as described below. More complex issues arise when opposite sides of the quadrilateral have unequal edge node distributions. In addition, rather than use the fluxes computed over the sides of the ir-

regular quadrilateral, we now compute the average velocity and pressure gradient over the entire control volume in the calculation of $\mathbf{k}^*$. This point will be discussed in more detail in section 6.2.4 below. We now consider each of these issues.

Figure 6-3: Triangulation of coarse cell illustrating resolution of cell geometry and internal heterogeneity

Figure 6-4: Mapping of general quadrilateral (physical space) to unit square (transform space)

In the case of an irregular quadrilateral with equal node distributions along opposite sides, periodicity in flux and pressure can be established via application of specifications similar to Eq. (6-4). However, the jump in pressure between an element edge on side 3 and the corresponding element edge on side 4 will not be the same for all locations along the quadrilateral side. This is due to the fact that the jump in pressure derives from the assumption that the global pressure is of the form $p = p_0 + \mathbf{G} \cdot \mathbf{x}$. Therefore, the jump in pressure for a general quadrilateral must be specified to be proportional to the distance between the corresponding points on sides 3 and 4 (and similarly for sides 1 and 2). In this case, for $G_x \neq 0$ and $G_y = 0$, Eqs. (6-4a) and (6-4b) will be replaced by:

$$p(x_1, y_1) = p(x_2, y_2) - G_x \Delta x_{12} \quad \text{on } \partial D_1 \text{ and } \partial D_2, \quad (6-9a)$$

$$p(x_3, y_3) = p(x_4, y_4) - G_x \Delta x_{34} \quad \text{on } \partial D_3 \text{ and } \partial D_4, \quad (6-9b)$$

where (x_j, y_j) , $j=1,4$ designates a point on side j , Δx_{12} is the difference in the x -locations of corresponding element edge midpoints on sides 1 and 2 and Δx_{34} is the analogous quantity for sides 3 and 4. Similar expressions apply when $G_y \neq 0$. The flux relationships (Eqs. (6-4c) and (6-4d)) do not require any modification in the case of equal distributions of nodes on opposite sides of the quadrilateral cell.

If the numbers of nodes on the opposite sides are different, or the lengths of the element edges on the opposite sides are different, a consistent interpolation of flux and pressure is required in order to specify periodicity properly. In this case, the correspondences between edges on opposite boundaries of the quadrilateral control volume must be established. We set these correspondences by viewing the domain boundaries in transform space. This is accomplished through use of an isoparametric mapping to transform the general quadrilateral into a unit square cell. This mapping is of the form:

$$\mathbf{x} = (1-\xi)(1-\eta)\mathbf{x}_1^v + \xi(1-\eta)\mathbf{x}_2^v + \xi\eta\mathbf{x}_3^v + (1-\xi)\eta\mathbf{x}_4^v, \quad (6-10)$$

where (ξ, η) are the coordinates in the transform space and vary between 0 and 1, and $\mathbf{x}_i^v$, $i=1, 4$ are the coordinates of the four vertices of the quadrilateral in physical space (see Fig. 6-4). The relative position of any point along the boundary remains unchanged after the transform, as is evident from Eq. (6-10).

Edge correspondence is now determined in transform space. We describe our treatment for pressure for an element edge located along side 3. In transform space, this edge lies along the $\xi=0$ side of the unit cell and spans in η from η_1 to η_2 . The corresponding region along side 4 ($\xi=1$) similarly includes all edges or portions of edges that span from η_1 to η_2 . Once the element edges on side 4 are determined, expressions equating the pressure at edges on side 3 to the pressures of the corresponding edges on side 4 can be established via Eqs. (6-9). These expressions are quite simple because we approximate pressure as piecewise constant along each element edge, which corresponds to the lowest-order consistent approximation for pressure.

The specification of periodicity in flux is more complicated. We again consider element edges located along sides 3 and 4. Once the element edges on side 4 corresponding to the edge of a particular element on side 3 are determined, expressions equating the fluxes through edges on side 3 to the corresponding edges on side 4 can be established. Flux through element edges is treated as piecewise constant, so portions of edges carry a fraction of the total edge flux equal to the fraction of their length relative to the edge length. Using this approach, conservative expressions for periodicity in flux can be built into the set of linear equations. See He (2000) for further details. We note that an alternate procedure is to treat the boundary conditions directly in Cartesian space, which gives rise to a “staircase” representation of the quadrilateral boundaries.

We have now described our procedure for the finite element solution of the single phase pressure equation subject to periodic boundary conditions over a general quadrilateral. We emphasize that, although we introduce the mapping (6-10) to allow us to apply periodic boundary conditions, our numerical solution is accomplished in physical (rather than transform) space. We now describe how the coarse scale cell permeability $\mathbf{k}^*$ is computed from our numerical solutions of Eq. (6-3) over the coarse cell region.

For the case of a rectangular coarse block, fluxes through the block sides can be computed and used directly in Eq. (6-5) for the determination of $\mathbf{k}^*$. For an irregular quadrilateral, however, the procedure to compute $\mathbf{k}^*$ is somewhat different. In this case we compute $\langle \mathbf{u} \rangle$ and $\langle \nabla p \rangle$ over the entire solution domain using an area-weighted averaging:

$$\langle \mathbf{u} \rangle = \frac{1}{A_q} \sum_{i=1}^{N_t} a_i \mathbf{u}_i, \quad (6-11)$$

where A_q is the area of the quadrilateral, a_i is the area of element i , $\mathbf{u}_i$ is the velocity computed at the centroid of element i and N_t is the total number of elements. A similar expression provides $\langle \nabla p \rangle$. In this case, due to numerical approximations, $\langle \nabla p \rangle$ is not identically equal to $\mathbf{G}$ (though they are very close). The equivalent grid block permeability $\mathbf{k}^*$ can now be computed via:

$$\langle \mathbf{u} \rangle = -\mathbf{k}^* \cdot \langle \nabla p \rangle. \quad (6-12)$$

The $\mathbf{k}^*$ computed in this way will not in general be symmetric. Symmetry is enforced by simply averaging the cross terms of $\mathbf{k}^*$.

The conforming scale up method described here for computing $\mathbf{k}^*$ has several positive features. The method provides results identical to those generated from existing procedures when the coarse block is rectangular and displays equal edge node distributions. In this case, $\langle \mathbf{u} \rangle$ computed using Eq. (6-11) is identical to that determined by integrating over the coarse block sides and $\langle \nabla p \rangle = \mathbf{G}$. In the case of a general quadrilateral control volume characterized by a constant (full tensor) permeability, the method provides an upscaled permeability equal to this constant value. Though this may appear to be an obvious consistency result, it is not always recovered if $\langle \mathbf{u} \rangle$ and $\langle \nabla p \rangle$ are not computed in an appropriate manner (e.g., this consistency may be lost if $\langle \mathbf{u} \rangle$ is computed by integrating over the coarse cell sides rather than via Eq. (6-11)). This completes our description of the upscaling procedure.

6.2.4 Alternative Approaches, Treatment of Polygonal Cells, and Gridding. Several other techniques for the calculation of $\mathbf{k}^*$ for general quadrilaterals were considered. An approach that gives results very close to those presented in the examples below entails the application of periodicity in pressure in transform space rather than in physical space. With this approach, the procedure to compute $\mathbf{k}^*$ for a general quadrilateral is exactly as described above except we set $\Delta x_{12}=0$ and $\Delta x_{34}=1$ in Eq. (6-9). This corresponds to specifying periodicity in pressure in transform space; i.e., over the unit square in Fig. 6-4. We then compute the general tensor $\mathbf{T}^*$, which accounts for both geometry and heterogeneity, rather than $\mathbf{k}^*$.

The general relationship between $\mathbf{T}$ and $\mathbf{k}$ is given by (e.g., Edwards and Rogers, 1998):

$$\mathbf{T} = \mathbf{J}^{-1} \mathbf{k} \mathbf{J}^{-T} \mathbf{D}, \quad (6-13)$$

where $J_{ij} = \partial x_i / \partial \xi_j$ is the Jacobian matrix of the coordinate transform, D is the determinant of $\mathbf{J}$, and the elements of $\mathbf{T}$ involve components of $\mathbf{k}$ and $\mathbf{J}$ (see Edwards and Rogers, 1998 for details). As viewed from the (ξ, η) transform space, the upscaling procedure generates an effective $\mathbf{T}$ (designated $\mathbf{T}^*$) rather than an effective $\mathbf{k}$ (Eek-Jensen et al., 1999; Edwards, 1996). For a general quadrilateral, with reference to Eq. (6-12), the relationship between $\langle \mathbf{u} \rangle$ and $\langle \nabla p \rangle$ provides $\mathbf{T}^*$ rather than $\mathbf{k}^*$. The effective Cartesian permeability tensor $\mathbf{k}^*$ can be computed from $\mathbf{T}^*$ by inverting Eq. (6-13):

$$\mathbf{k}^* = \frac{1}{D} \mathbf{J} \mathbf{T}^* \mathbf{J}^T . \quad (6-14)$$

However, in terms of providing upscaled properties in physical space, this approach has the drawback that it does not always produce the result $\mathbf{k}^* = \mathbf{k}$ in cases where the underlying cell permeability is homogeneous. This is related to the fact that a unique Jacobian matrix can be obtained only when the cell is a parallelogram, in which case the transform derivatives are constants and exact results are obtained. For general quadrilaterals, $\mathbf{J}$ varies with position and there is no unique way to calculate $\mathbf{J}$. The derivatives must therefore be approximated, which introduces some error in computing $\mathbf{k}^*$. This makes an assessment of the accuracy of the method more difficult when comparing against other techniques that generate $\mathbf{k}^*$ directly. We note that $\mathbf{T}^*$ can be used directly in coarse scale models (Eek-Jensen et al., 1999; Edwards, 1996) and we plan to explore this approach in future work.

A separate issue involves the types of boundary conditions used in the calculation of $\mathbf{k}^*$. Our formulation is general in this regard, as any type of boundary conditions can be applied within the general framework. Although no such investigation was performed in this work, it is quite likely that boundary specifications other than periodicity may provide more accurate grid block permeabilities in some cases, particularly when the coarse cell is highly distorted. Since the conforming scale up method accurately treats geometric effects, we can use it to isolate, and thereby assess, the effects of various boundary specifications. This could allow for the determination of optimal boundary specifications for the calculation of $\mathbf{k}^*$.

The procedure described here can be readily extended to handle any type of two dimensional polygonal coarse cell. Additional considerations that must be addressed in this case are the triangulation of the general two dimensional control volume, which is straightforward, and the im-

sition of boundary conditions for the solution of the local problem. Because our treatment of boundary conditions described above assumes that the coarse cell contains four distinct “sides,” some generalization of the procedure will be required. One approach is simply to impose pressure boundary conditions linear in x for one solution (i.e., $p=x$) and boundary conditions linear in y for the second solution. Then, $\mathbf{k}^*$ could be computed directly through the use of Eqs. (6-11) and (6-12).

Grid quality is also an important issue for any numerical method and our case is no exception. As discussed above, flow-based grids have been previously developed within the context of reservoir simulation by a number of researchers (e.g., Tran, 1995; Wen, 1996; Verma and Aziz, 1997; Edwards et al., 1998; Castellini et al., 2000). In one of the examples below, we apply the conforming scale up method in conjunction with a flow-based gridding procedure. The flow-based grid in this case is generated by solving a global flow problem and contouring streamlines and isopotentials (as described in Chapter 3). We select a number of these streamline and isopotential lines and then perform a Laplacian smoothing to improve the grid quality. This provides a flow-based grid that resolves important flow effects but is not overly concentrated in high flow regions. The general grid generation procedure is described in more detail by Wen et al. (2001).

6.3. Numerical Results Using the Conforming Scale Up Method

In this section we present a variety of numerical calculations using the conforming scale up method. Results are presented for an isolated, layered parallelogram, for which exact results are available, and for global flow through systems discretized using coarse irregular quadrilaterals.

We note first that, as indicated above, in cases where the fine scale permeability within the coarse cell is homogeneous, our method reproduces this permeability exactly. This result was verified numerically using diagonal and full tensor permeabilities and irregular quadrilaterals. This basic consistency follows from our procedure for computing $\mathbf{k}^*$, as can be demonstrated from Eqs. (6-6), (6-8), (6-11) and (6-12).

We now consider the case of a four-layer system to be upscaled to a parallelogram coarse grid cell, as shown in Fig. 6-5. Layer permeabilities are isotropic and are given by $k_1=1$, $k_2=5$, $k_3=20$, $k_4=50$. In the upscaling calculation, nodes are aligned with the layer boundaries. The equivalent permeability for this system can be computed exactly and is a diagonal tensor with

$k_{xx}^* = 19$ and $k_{yy}^* = 3.15$. This result is recovered exactly by our method, verifying the consistency of the approach.

The next example again involves a four layer system with isotropic permeabilities $k_1 - k_4$, as shown in Fig. 6-6. Permeability values here are $k_1 = 0.1$, $k_2 = 1$, $k_3 = 10$, $k_4 = 100$. In this case, however, the global system (a unit square) is to be solved using the 5×5 system of irregular quadrilaterals depicted in the figure. These quadrilaterals were generated by randomly placing nodes along the sides of the global domain. As a result, the coarse cells are not aligned with the layer boundaries, so the cell permeabilities must be computed from the underlying permeability description. We compute coarse cell permeabilities for each of the 25 cells and then solve a global flow problem with $p=1$ along the left face, $p=0$ along the right face and no flow along the upper and lower faces. This global flow problem can be solved using either the finite element method described in this chapter (with each quadrilateral divided into two triangular elements) or using a flux-continuous finite volume method (Edwards and Rogers, 1998).

The analytical solution for the flow rate in this case, designated q_x , is 27.8. For flow in the y -direction ($p=1$ along the lower face, $p=0$ along the upper face and no flow along the lateral faces) the analytical solution gives $q_y = 0.360$. Applying our procedure, we obtain $q_x = 27.5$ and $q_y = 0.373$. These results are quite accurate (errors of a few percent or less) and indicate that the up-scaling procedure is able to accurately resolve subgrid heterogeneity in general quadrilaterals.

Figure 6-5: Parallelogram cell with layered permeability field

Figure 6-6: Layered system with 5×5 grid of irregular quadrilaterals

We now assess the performance of our upscaling method for cells of varying geometry through a comparison with global flow results computed on both fine and uniform coarse grids. This case involves flow through a geostatistical permeability field (Fig. 6-7), generated using the GSLIB (Deutsch and Journel, 1998) algorithms. The domain is again a unit square and the fine grid is of dimensions 50×50. The permeability field is characterized by a correlation length in the x -direction of 0.5 and a correlation length in the y -direction of 0.1. Permeability is log-normally distributed, with the variance of $\log k$ equal to 4. We set $k_y=k_x$; i.e., permeability is isotropic on the fine scale.

This system is then upscaled to 10×10 grids of the type shown in Fig. 6-8. We consider three such 10×10 grids, each of which is generated by randomly placing nodes along side boundaries, as described above. Note that the upscaled permeability field shown in Fig. 6-8 resembles the underlying fine scale permeability field to some extent (Fig. 6-7), as would be expected. Due to

the random nature of the coarse grid structure, however, it is clear that the grid does not accurately resolve the extremes in permeability evident in the fine scale description.

Global flow results for the fine and coarse grids are computed for flow in both the x - and y -directions. These fine grid solutions give $q_x = 2.59$ and $q_y = 0.820$. We then upscale this fine grid model to a 10×10 uniform coarse grid and solve the global flow problem on this grid to provide the reference solution. The flow results for the uniform coarse grid are $q_x = 2.46$ and $q_y = 0.754$.

Figure 6-7: Heterogeneous permeability field with correlation length of 0.5 in the x -direction and 0.1 in the y -direction

Figure 6-8: Irregular 10×10 quadrilateral grid and upscaled permeability field

Flow results on the irregular 10×10 quadrilateral grids are as follows. For flow in the x -direction, the three grids give results for q_x of 2.38, 2.27, and 2.25. For flow in the y -direction, we obtain values of q_y of 0.734, 0.698 and 0.749. These results are less accurate than the uniform coarse grid results, as would be expected since the cells vary significantly in size and some of them are quite irregular. However, the general level of accuracy of the results (within 15% of the fine grid results for q_x and q_y) is still quite reasonable. This is further illustrated in Fig. 6-9, where we display pressure contours for the fine and coarse scale results (the coarse scale results correspond to the grid shown in Fig. 6-8). Here, the pressure contours show a general correspondence, though differences are apparent, particularly in the high permeability regions of the domain. We note finally that, in practice, the grid will be refined in important (e.g., high flow) regions and coarse in less important regions. Thus, errors due to grid irregularity will be more than compensated for by the enhanced resolution of key aspects of the permeability field.

We now illustrate the impact of local grid resolution by applying the conforming scale up method in conjunction with a flow-based grid. The underlying permeability field, shown in Fig.

6-10, is of dimension 100×100 and contains layering oriented at an angle of 30° relative to the x -axis. The correlation length along the layering is 0.8 while the correlation length normal to the layering is 0.04 (the domain is again a unit square). Permeability is log-normally distributed, with the variance of $\log k$ equal to 4. The basic field was again generated using GSLIB (Deutsch and Journel, 1998). In each fine scale block we set $k_2 = 0.1k_1$, where k_1 is the principal value of permeability in the direction along the layering and k_2 is the principal value of permeability in the direction across the layering. In the x - y coordinate system, the fine scale $\mathbf{k}$ is a full tensor quantity. The 100×100 grid was upscaled to both a 20×20 uniform grid and a 20×20 flow-based grid (Fig. 6-11). The flow-based grid was generated using the procedure described above. Permeability tensors for each coarse cell in the flow-based grid were computed using the conforming scale up method.

Figure 6-9: Pressure contours ($p = 0.9, 0.8, 0.7, \dots, 0.1$) for the fine (solid curves) and coarse (dotted curves) solutions. The coarse solutions correspond to the grid and permeability field of Fig. 6-8

Figure 6-10: Layered permeability field oriented at an angle of 30° to the x -axis with correlation length of 0.8 along the layering and 0.04 normal to the layering

Figure 6-11: Flow-based grid for permeability field of Fig. 6-10

Simulations were performed for unit mobility ratio displacements. In this case water is assumed to have properties identical to that of the displaced oil. The governing equation for the displacement is:

$$\frac{\partial S}{\partial t} + \nabla \cdot (\mathbf{u} S) = 0, \quad (6-15)$$

where S is water saturation (volume fraction). Once the pressure field is determined, as described above, the velocity field $\mathbf{u}$ is computed through application of Darcy's law. Streamlines are then mapped by tracing the velocity field and Eq. (6-15) solved by integrating along streamlines. The procedure for streamline generation and the subsequent integration of Eq. (6-15), described by Prevost et al. (2001), accurately accounts for the geometric irregularity of the quadrilateral cells and the full tensor nature of the permeability field.

In the simulations, water is injected at the left ($x=0$) boundary and production is taken at the right boundary. Flow results for water cut (denoted by F_w and defined as the fraction of water in the produced fluid) for the three cases are shown in Fig. 6-12. The solid curve represents the fine scale results, the dotted curve the 20×20 uniform coarse scale results and the dashed curve the results using the conforming scale up method (CSUM) with the 20×20 flow-based grid. The uniform coarse grid provides results in significant error and overpredicts the breakthrough time for water by about a factor of two. The flow based grid, by contrast, predicts an accurate breakthrough time and provides results in close agreement with the reference fine scale results over the entire course of the run.

The results for total flow rate (q_x) are also more accurate using the conforming scale up method with the flow-based grid. For the fine grid we obtain $q_x=0.64$. The uniform coarse grid gives $q_x=0.50$ and the flow-based grid gives $q_x=0.63$. Thus, the method provides considerably improved results for both F_w and q_x in this case. We note finally that, because the full tensor aspect of permeability and the geometric effect of the irregular quadrilaterals are properly treated, the differences between the coarse scale results are due to the effects of the grid and the cell permeability calculation.

Figure 6-12: Simulation results for water cut for fine model (solid curve), uniformly coarsened 20×20 model (dotted curve), and flow-based 20×20 model with cell permeabilities computed using conforming scale up method (dashed curve)

6.4. Discussion and Conclusions

The use of general curvilinear grid generation techniques leads to coarse grid cells that are non-aligned with respect to the underlying geological grid. Standard upscaling techniques are therefore not directly applicable for such problems. In this paper, we introduced a new approach for the calculation of upscaled permeabilities for general quadrilateral grid cells, which we call the conforming scale up method. The method proceeds by triangulating the coarse scale quadrilateral such that internal heterogeneity and cell geometry are accurately resolved. The solution of the local single phase pressure equation, subject to periodic boundary conditions, and the subsequent averaging of the fine scale solution provides the cell permeability ($\mathbf{k}^*$).

The conforming scale up method was applied to a number of cases involving layered and geostatistical heterogeneity fields. In all cases considered the method provided results of reasonable accuracy. When used in conjunction with a flow-based grid, results using the conforming scale up method were considerably more accurate than those obtained using a uniform coarse

grid. This demonstrates the overall applicability of the combined use of conforming upscaling and flow-based gridding techniques.

The formulation presented here can be combined with other recent upscaling developments to provide further improvements. For example, our approach could be used in conjunction with the formulation of Zijl and Trykozko (2001) to force the convergence of the cell permeabilities. In the examples presented in this paper, the number of elements in the local problem is determined solely by the number of blocks characterizing the subgrid permeability field. However, as shown in Zijl and Trykozko (2001), more accurate estimates for k^* can be obtained through further grid refinement or through the use of both mixed and nodal finite element methods. In addition, the use of border regions around the target cell, as applied by Wen et al. (2000) for Cartesian grids, might improve the accuracy of the cell permeability tensors. The effect of local boundary conditions is also an important issue that should be better quantified through further investigation. Finally, the extension of our method to the three dimensional case will provide even greater applicability. Efforts along these lines are currently underway.

Acknowledgments

We are grateful to Dr. X.H. Wen (Chevron Petroleum Technology Company) for providing us with the permeability field and flow-based grid used in the final example and to M. Prevost (Stanford University) for providing us with his streamline simulation code.

6.5 References

- Barber, C.B., Dobkin, D.P. and Huhdanpaa, H.: "The Quickhull Algorithm for Convex Hulls," *ACM Transactions on Mathematical Software*, **22**, 469-483, 1996.
- Boe, O.: "Analysis of an Upscaling Method Based on Conservation of Dissipation," *Transport in Porous Media*, **17**, 77-86, 1994.
- Castellini, A., Edwards, M.G. and Durlofsky, L.J.: "Flow Based Modules for Grid Generation in Two and Three Dimensions," Proceedings of the 7th European Conference on the Mathematics of Oil Recovery, Baveno, Italy, Sept. 5-8, 2000.
- Deutsch, C.V. and Journel, A.G.: *GSLIB: Geostatistical Software Library and User's Guide*, 2nd edition, Oxford University Press, New York, 1998.
- Durlofsky, L.J.: "Numerical Calculation of Equivalent Grid Block Permeability Tensors for Heterogeneous Porous Media," *Water Resour. Res.*, **27**, 699-708, 1991.
- Durlofsky, L.J., Jones, R.C. and Milliken, W.J.: "A Nonuniform Coarsening Approach for the Scale-Up of Displacement Processes in Heterogeneous Porous Media," *Adv. Water Resour.*, **20**, 335-347, 1997.
- Edwards, M.G.: "Superconvergent Renormalization and Tensor Approximation," Proceedings of the 5th European Conference on the Mathematics of Oil Recovery, Z.E. Heinmann, ed., Leoben, Austria, Sept. 3-6, 1996.
- Edwards, M.G., Agut, R. and Aziz, K.: "Quasi K-Orthogonal Streamline Grids: Gridding and Discretization," paper SPE 49072 presented at the SPE Annual Technical Conference and Exhibition, New Orleans, Sept. 27-30, 1998.
- Edwards, M.G. and Rogers, C.F.: "Finite Volume Discretization with Imposed Flux Continuity for the General Tensor Pressure Equation," *Comp. Geosciences*, **2**, 259-290, 1998.
- Eek-Jensen, C.F., Aavatsmark, I. and Boe, O.: "Upscaling on General Quadrilateral Grids in 3D with Application to Field Cases," Proceedings of the EAGE - 10th European Symposium on Improved Oil Recovery, Brighton, UK, Aug. 18-20, 1999.
- He, C.: "Upscaling to General Quadrilateral Grids," Master's report, Stanford University, 2000.
- Marini, L.D.: "An Inexpensive Method for the Evaluation of the Solution of the Lowest Order Raviart-Thomas Mixed Method," *SIAM J. Numer. Anal.*, **22**, 493-496, 1985.
- Pickup, G.E., Ringrose, P.S., Jensen, J.L. and Sorbie, K.S.: "Permeability Tensors for Sedimentary Structures," *Math. Geol.*, **26**, 227-250, 1994.
- Prevost, M., Edwards, M.G. and Blunt, M.J.: "Streamline Tracing on Curvilinear Structured and Unstructured Grids," paper SPE 66347 presented at the SPE Reservoir Simulation Symposium, Houston, Feb. 11-14, 2001.

- Renard, P. and de Marsily, G.: "Calculating Equivalent Permeability: A Review," *Adv. Water Resour.*, **20**, 253-278, 1997.
- Tran, T.T.: "Stochastic Simulation of Permeability Fields and their Scale Up for Flow Modeling," Ph.D. thesis, Stanford University, 1995.
- Trykozko, A., Zijl, W. and Bossavit, A.: "Nodal and Mixed Finite Elements for the Numerical Homogenization of 3D Permeability," preprint provided by the authors (2000).
- Verma, S. and Aziz, K.: "A Control Volume Scheme for Flexible Grids in Reservoir Simulation," paper SPE 37999 presented at the SPE Reservoir Simulation Symposium, Dallas, June 8-11, 1997.
- Wen, X.H.: "Stochastic Simulation of Groundwater Flow and Mass Transport in Heterogeneous Aquifers: Conditioning and Problems of Scale," Ph.D. thesis, Polytechnic University of Valencia, Spain, 1996.
- Wen, X.H., Durlofsky, L.J. and Edwards, M.G.: "Upscaling of Channel Systems in Two Dimensions," in review for *Transport in Porous Media*, 2001.
- Wen, X.H., Durlofsky, L.J., Lee, S.H. and Edwards, M.G.: "Full Tensor Upscaling of Geologically Complex Reservoir Descriptions, paper SPE 62928 presented at the SPE Annual Technical Conference and Exhibition, Dallas, Oct. 1-4, 2000.
- Wen, X.H. and Gomez-Hernandez, J.J.: "Upscaling Hydraulic Conductivities in Heterogeneous Media: An Overview," *J. Hydrology*, **183**, ix-xxxii, 1996.
- Zijl, W. and Trykozko, A.: "Numerical Homogenization of the Absolute Permeability using the Conformal-Nodal and the Mixed-Hybrid Finite Element Method," *Transport in Porous Media*, to appear 2001.