PRODUCTIVITY AND INJECTIVITY OF HORIZONTAL WELLS

Annual Report
March 10, 1997-March 9, 1998

By
Khalid Aziz
Thomas A. Hewett
Sepehr Arbabi
Marilyn Smith

Date Published: November 1999

Work Performed Under Contract No. DE-FG22-93BC14862

Stanford University
Stanford, California

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.

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.

DOE/BC/14862-23
Distribution Category UC-122

Productivity and Injectivity of Horizontal Wells

By
Khalid Aziz
Thomas A. Hewett
Sepehr Arbabi
Marilyn Smith

November 1999

Work Performed Under Contract No. DE-FG22-93BC14862

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

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

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

Contents

1	Flexible Streamline Grids for Reservoir Simulation	1
1.1	Introduction	1
1.2	FLEX Simulator	2
1.3	Flexible Grids	3
1.4	Flow Equations	3
1.5	Numerical Schemes	4
1.6	Flow in a Channel with High Curvature	6
1.6.1	Boundary Aligned Grid (BAG-Grid)	6
1.6.2	Local Modular Refinement	8
1.6.3	Results	8
1.7	Streamline Grids and Discretization	10
1.8	General Curvilinear Coordinates	11
1.9	Quadrilateral Discretization	12
1.9.1	Numerical Scheme	12
1.9.2	(a) Cell-centered: Transform Space Flux Continuity	13
1.9.3	(b) Cell-centered: Physical Space Flux Continuity	14
1.9.4	(c) Point Distributed: Physical Space Flux Continuity	15
1.10	Results	15
1.10.1	Heterogeneous Field	17
1.10.2	Meandering Homogeneous Channel	19
1.10.3	Meandering Heterogeneous Channel	19
1.11	Conclusions and Future Work	25
2	Well Index Correlation for a Finite Conductive Horizontal Well	31
2.1	Introduction	31
2.2	Conventional Well Model	32
2.3	Semi-Analytical Model	33

2.4	Well Pressure Drop	34
2.5	Comparative Simulation Study	35
2.5.1	Constraints of the Analytical Model	37
2.5.2	Correct Block Pressure	38
2.5.3	Calculation of m	38
2.5.4	Cases: Well Index Analysis and Correction	40
2.5.5	Impact of other parameters	42
2.6	Conclusions	44
3	Analysis of Gas Injection in Horizontal Wells	53
3.1	Introduction	53
3.2	Literature Survey	53
3.3	Viscous and Gravitational Forces	55
3.4	Experimental Setup and Procedure	55
3.5	Simulation of the experiment	56
3.6	Sensitivity Analysis	58
3.7	Expeimental Results	63
3.8	Matching Experimental Data with Simulation	63
3.9	Conclusions	70
4	Multiple Solutions for Separated Gas-Liquid Flow in a Wellbore	81
4.1	Introduction	81
4.2	Occurrence of Multiple Solutions	82
4.3	Parameters Affecting Multiple Solutions	83
4.4	Multiple Solution Region: X - Y Map	86
4.5	Effects of Wall Inflow/Outflow	90
4.6	Multiple Solution Analysis via Interfacial Friction Shear	95
4.7	Physical Validation of Multiple Solutions	99
4.8	Numerical Validation of Multiple Solutions	102
4.9	Multiple Solutions for Annular-Mist Flow	106
4.10	Concluding Remarks	114

5	An Experimental Study of Single Phase and Two-Phase Flow in Horizontal Wells	117
5.1	Introduction	117
5.2	Experimental Setup and Types of Experiments	118
5.3	Analysis of Experimental Data	120
5.3.1	Wellbore Calibration Experiment	120
5.3.2	Single-phase Water Wellbore Flow	121
5.3.3	Two Phase Wellbore Flow	121
5.4	Prediction by Existing Models	123
5.4.1	Pipe Roughness Determination	123
5.4.2	Analytical Procedure	124
5.4.3	Discussion of Results	125
5.5	Development of New Wellbore Flow Models	126
5.6	Summary and Observation	126
6	A Mechanistic Model for Multiphase Flow in Pipes	139
6.1	Introduction	139
6.2	Flow Pattern Determination	140
6.2.1	Dispersed Bubble Flow	140
6.2.2	Stratified Flow	141
6.2.3	Annular-Mist Flow	143
6.2.4	Bubble Flow	145
6.2.5	Intermittent Flow	145
6.3	Calculation of Pressure Drop and Liquid Volume Fraction	147
6.3.1	Dispersed Bubble Flow	147
6.3.2	Stratified Flow	148
6.3.3	Annular-Mist Flow	148
6.3.4	Bubble Flow	148
6.3.5	Intermittent Flow	149
6.3.6	Froth Flow	150
6.4	Results	151

6.5	Comments	152
6.6	Conclusions	153
7	A Comprehensive Reservoir/Wellbore Model for Horizontal Wells	163
7.1	Introduction	163
7.2	Reservoir Model	164
7.3	Infinite-Conductivity Well Model	165
	7.3.1 Example Problem	167
	7.3.2 Applications of Infinite Conductivity Wellbore Model	168
7.4	Finite-Conductivity Well Model	168
7.5	Testing the Models	171
	7.5.1 Transient Effects	172
	7.5.2 Pseudosteady State	173
7.6	Effect of Directional Permeabilities	174
7.7	Constant Drawdown at Well Heel	175
7.8	Running the Models with Other Constraints	175

Management Summary

1. DOE Approval

The DOE approval for the annual renewal of the research grant to the Stanford Project on the *Productivity and Injectivity of Horizontal Wells* was received in late March 1997. The Project formally commenced on March 10, 1993. Mr. Thomas Reid is the DOE Project Manager in Bartlesville and Mr. John Augustine is the DOE Contracts Officer in Pittsburgh.

2. Industrial Affiliates Program

The DOE Project operates in association with an Industrial Affiliates program on horizontal wells, for which oil company membership has also continued during 1996. The current membership comprised the following organizations:

AGIP (Italy)
Amoco (USA) ARTEP/IFP (France)
BP Exploration (USA)
Chevron (USA)
INTEVEP (Venezuela)
Marathon (USA)
Maxus Energy (USA)
Mobil E & P (USA)
Norsk Hydro (Norway)
Petrobras America (Brazil)
Phillips Petroleum (USA)
Shell Exploration (USA)
Smedvig Technologies (Norway)
Texaco (USA)
Union Pacific Resources (USA)
US Department of Interior/
Minerals Management Service (USA)

3. Project Goals

The Project has eight principal goals to be studied and developed over a five year period. These goals are as follows:

- TASK 1:** Advanced Modeling of Horizontal Wells - Develop special gridding techniques and associated averaging algorithms for accurate simulation of HW-performance.
- TASK 2:** Investigate and Incorporate the Effects of Reservoir Heterogeneities - Study impacts of various types of heterogeneity and develop methods for incorporating their effects in both fine grid and coarse grid models.
- TASK 3:** Develop Improved Methods for Calculating Multi-Phase Pressure Drops within the Wellbore - Plan, execute, and interpret two-phase flow experiments at an oil company research facility, and use results to analyze/validate a new two-phase model.
- TASK 4:** Pseudo-Functions - Define improved methods for computing two-phase pseudo-functions for effective relative permeabilities for coarse grid blocks near an HW - determine sensitivities to heterogeneities, flow conditions, skin factors, etc.
- TASK 5:** Develop Multi-Well Models - Develop numerical techniques and software in a parallel computing architecture capable of interactively coupling multiple detailed HW-models to a large scale reservoir simulator.
- TASK 6:** Test HW-Models with Field Examples - Work with affiliate's member companies to establish HW-modeling capabilities from field measurements, particularly for pathological problem cases.
- TASK 7:** EOR Applications - Provide and implement practical HW aspects into modeling of EOR processes - miscible gas, steam displacement, in-situ combustion.
- TASK 8:** Application Studies and Their Optimization - Seek field opportunities for HW's and study their best implementation in various reservoir scenarios e.g., multiple laterals, hydraulic fracture variants, etc.

Some of our principal achievements during 1997 are summarized below.

4. Flexible Streamline Grids for Reservoir Simulation (Chapter 1)

Conventional simulators using Cartesian grids are generally unable to provide a realistic representation of critical reservoir features such as faults, fractures, and channels. Flexible grids which honor curved geometries can model complex reservoirs more effectively. Since streamlines provide a natural means of identifying crucial flow regions, they can be

used to define the base grids in a flexible grid approach. A new grid generation technique based on streamlines and equipotentials is developed for reservoir simulation. The use of full tensor permeabilities ensures that a consistent approximation is retained for any grid type. Results computed using the new grids demonstrate improvement in accuracy when compared with Cartesian grids of similar resolution.

5. Well Index Correlation for a Finite Conductive Horizontal Well (Chapters 2)

Previously we have shown that the correct well index for a typical partially penetrating horizontal well could be very different from the classical Peaceman well index. The correct well index can be calculated in a semi analytical manner in which the well pressure is obtained analytically and the well block pressure is computed numerically. The effect of pressure drop (frictional and accelerational) in a horizontal well on well index is studied on basis of well indices. Correct well indices are computed for finite conductivity wells. Their use in simulations are shown to improve the numerical results. Several example are included.

6. Analysis of Gas Injection in Horizontal Wells (Chapter 3)

Gravity is a major force affecting the recovery of oil from reservoirs by gas injection. For vertical wells, there are severe rate limitations to achieve the gravity drainage benefits, but the use of longer horizontal wells may allow this technique to be economically attractive. The effect of gas injection on the gravity drainage process in horizontal wells is studied through experiments and simulations. The sweep efficiency of the process for various well configurations has been studied using a CT scanner. A sensitivity study on the impacts of many parameters such as length and location of wells, injection rate, and permeability on recovery efficiency has also been carried out.

7. Multiple Solutions for Separated Gas-Liquid Flow in a Wellbore (Chapter 4)

Many researchers have reported the existence of multiple solutions to the momentum balance equation for separated upward gas-liquid two-phase flow, such as stratified smooth flow, stratified wavy flow, annular flow, and annular mist flow. This phenomena is examined in detail in this chapter. The size of multiple solution region and its dependence on flow parameters are investigated. Physical arguments and numerical simulations of a transient process show that the smallest root provides the correct solution.

8. An Experimental Study of Single Phase and Two-Phase Flow in Horizontal Wells (Chapter 5)

This chapter contains a brief overview of the 5-year Stanford horizontal wellbore experimental program. Comparison between measurements and predictions from different existing mechanistic and empirical flow models are reported. It is concluded that none of the existing models can make reliable predictions for pressure drops for two-phase wellbore flow. Application of a new simplified homogeneous model shows promising results.

9. A Mechanistic Model for Multiphase Flow in Pipes (Chapter 6)

Advances have been made in the Stanford mechanistic flow model. A new mechanistic model, applicable to all pipe geometries and fluid properties is presented. The model incorporates roughness effects as well as liquid entrainment, both of which are not accounted for by previous models. New empirical correlations are proposed for liquid/wall and liquid/gas interfacial friction in stratified flow, for the liquid fraction entrained and the interfacial friction in annular-mist flow. Through extensive testing, the model has proven to be more robust than existing models and is applicable over a wide range of conditions.

10. A Comprehensive Reservoir/Wellbore Model for Horizontal Wells (Chapter 7)

While simulators can be used to predict the performance of horizontal wells, simple models are needed to quickly respond to such questions as: what is the effect of wellbore pressure drop and various reservoir and fluid parameters on well performance? To answer such questions, three-dimensional, anisotropic and transient, reservoir/wellbore coupling models for infinite-conductivity and finite-conductivity wellbores are developed. The finite-conductivity well model considers frictional and accelerational pressure drops in the wellbore and also takes into account the effects of fluid inflow into the well. These models can provide reference solutions for well productivity or well index and can be used to check the accuracy of numerical simulators in single phase flow model.

11. Affiliates Progress Review Meeting

An annual review meeting for member companies in the Stanford Horizontal Well Project was held at Stanford on October 27-28, 1997. This meeting was well attended and some member companies made presentations related to this project on the second day.

12. Papers and Presentations at Conferences, etc.

Seven papers were presented:

1. *"Simple Equations for Wellbore Pressure Drawdown Calculation"* presented at the 1997 SPE Western Regional Meeting - Long Beach, California, June 25-27
2. *"A Practical Procedure to Predict Cresting Behavior in Horizontal Wells"* presented at the Fifth Latin American and Caribbean Petroleum Engineering Conference and Exhibition, 1997 - Rio de Janeiro, Brazil, August 30 - September 3
3. *"Modeling the Behavior of Horizontal Wells"* presented at the 1997 IEA Workshop and Symposium on EOR - Copenhagen, Denmark, August 31 - September 3
4. *"A Comprehensive Reservoir/Wellbore Model for Horizontal Wells"* presented at the 1998 SPE India Oil and Gas Conference - New Delhi, India, April 7-9
5. *"A Mechanistic Model for Multiphase Flow in Pipes"* presented at the 49th Annual Meeting of CIM, 1998 - Calgary, Canada, June 8-10
6. *"An Experimental Study of Single Phase and Two-Phase Flow in Horizontal Wells"* presented at the 1998 SPE Western Regional Meeting - Bakersfield, May 10-13
7. *"Effects of Grid Systems on Predicting Horizontal Well Productivity"* presented at the 1998 SPE Western Regional Meeting - Bakersfield, May 10-13

Results of our research were presented to the following organizations or meetings:

- SPE Asia Pacific Oil and Gas Conference, Kuala Lumpur, Malaysia, April 14-16, 1991
- DOE Conference, Houston, June 16-18, 1997
- British Petroleum, London, UK, August 4-6, 1997
- Norsk Hydro, Bergen, Norway, August 25-26, 1997

1. Flexible Streamline Grids for Reservoir Simulation

by Régis Augut, Michael G. Edwards, and Khalid Aziz

Abstract

Critical reservoir features such as faults, fractures and highly correlated channels can have a high degree of geometric complexity with regions containing very high curvatures within their description. Conventional simulators are generally unable to provide a realistic representation of such features.

Flexible grids that honor curved geometries can enable geometrically and geologically complex reservoirs to be modeled effectively provided that a consistent flux approximation is employed. The generation of such grids and the development of accompanying appropriate flux approximations is a major area of current simulation research.

This chapter addresses both the generation of suitable simulation grids for heterogeneous media and specific discretization issues that arise with such grids. Streamlines and equipotentials are used to define our base grids. Since streamlines are concentrated in high velocity regions they provide a natural means of clustering fine grid cells in crucial flow regions.

For complex configurations and particularly for strongly heterogeneous regions the resulting grid cells can become very distorted due to extremely high curvatures. Two types of cell centered formulation are examined together with a cell vertex-point distributed scheme. Important distinctions are found for highly distorted cells.

The new grids are tested for accuracy in terms of critical breakthrough parameters and it is shown that a much higher level of grid resolution is required by conventional simulators in order to achieve results that are comparable with those computed on relatively coarse streamline-potential grids.

1.1 Introduction

Reservoir simulation involves the application of numerical methods to solve a system of conservation laws that describe flow in a porous medium. The results of the simulation process are strongly dependent upon the computational grid, geometry representation, and the accuracy of the discretization scheme.

The most appropriate grid tends to be problem dependent. Commonly used Cartesian grids are generally non-aligned with reservoir heterogeneities and consequently can provide poor representation of effective properties.

The development of a new object oriented reservoir simulator for handling flexible grids (called FLEX) is described in [1, 2]. Different types of grids and control-volumes are considered such as Voronoi, Control Volume Finite Element (CVFE). Streamline-based grids have also been used in FLEX but only in limited cases.

In this work a new grid generator based on streamlines and equipotential lines is developed, in order to improve the accuracy of reservoir simulation in heterogeneous media.

In the next Section a summary of features in FLEX is provided together with a brief description of the finite volume schemes employed. An example of a reservoir with a high permeability channel is used to show the advantages of both boundary aligned grids and grids defined by modular refinement.

The new method of grid generation based on streamlines and equipotentials is described next. A summary of issues concerning discretization on general curvilinear grids is also presented. The definition of the control-volume is shown to play a key role in defining the generalised flux on highly distorted grids.

Finally the results presented demonstrate the benefits of using streamline-potential grids. The natural ability of the method to generate grids that concentrate grid lines in critical high-flow regions of the reservoir is demonstrated. Flow simulation results computed on streamlines show that they provide the ability to compute critical flow parameters accurately while using only a small fraction of grid blocks required by conventional simulation in order to achieve the same resolution.

1.2 FLEX Simulator

The FLEX simulator is an object oriented, flexible grid, (two phase) black-oil reservoir simulator. The object oriented approach is particularly useful because of the complexity of grid generation and flexible grid geometries (such as Voronoi, CVFE, Hexagonal grids, etc.). All the developments have therefore been made following this approach using the C++ language.

1.3 Flexible Grids

Flexible grids are generally comprised of polygons whose shape and size can vary according to position in the reservoir. Such grids overcome the geometrical limitation of commonly used Cartesian grids.

The types of grid and the corresponding numerical model implemented in FLEX offer a much broader range of geometrical and geological application.

In particular FLEX is able to align grids with :

- Reservoir boundaries
- Beds with varying permeability tensors (FLEX accepts full and asymmetric tensors)
- Faults
- Horizontal/Inclined wells

Several modules (as originally proposed in [3]) can be assembled within FLEX to generate a desired grid. The modules can be scaled, rotated and placed in any combination at particular locations (in the global coordinate system) in the reservoir using simple coordinate transformations. The module options include Cartesian, Hexagonal, Parallel Cartesian, Tilted hexagonal or Radial.

Apart from reservoir grids that are entirely logically rectangular, the grids produced by the modular approach adopted in FLEX are generated by passing the final set of grid nodes defined by combinations of various modules to a Delaunay triangulator. For point distributed schemes permeabilities are assigned to the resulting control volumes, which are generally polygons. Further details of FLEX and the modular grids can be found in [1].

1.4 Flow Equations

Without loss of generality in the approach, we consider two phase flow in a porous media. The continuity equation for the j^{th} phase is written as

$$\int_{\Omega} \left(\frac{\partial(\phi s_j)}{\partial t} + \nabla \cdot \mathbf{v}_j \right) d\tau = M_j \quad (1.1)$$

where the integral is taken over domain Ω . The j^{th} phase saturation is defined by the ratio of phase volume τ_j to pore volume τ_{pv} with $s_j = \tau_j / \tau_{pv}$, M_j is a specified phase flux and ϕ is the porosity. Since the pore volume must always be filled by the fluids present, this gives

rise to the equation of state $\sum_{j=1}^{N_p} s_j = 1$. The usual momentum equations of fluid dynamics are replaced by Darcys law where

$$\mathbf{v}_j = -\lambda_j \mathbf{K} \nabla \Phi \quad (1.2)$$

is the j^{th} phase velocity, $\mathbf{K}$ can be a full rock permeability tensor, Φ is the pressure and the mobility is given by $\lambda_j = k_{rj}(s_j)/\mu_j$ where μ_j and k_{rj} are the respective phase viscosity and relative permeability. Gravity is neglected in this report. Further details on flow equations can be found in [4] and [5].

1.5 Numerical Schemes

The numerical schemes considered here are either cell centered or general point distributed approximations where flow variables, permeability and porosity are assumed to have a piecewise constant variation over each control volume. Continuity of flux and pressure are physical properties of the single phase flow equation that must be respected by the numerical approximation scheme. In one dimension it can be shown that continuous flux and pressure constraints lead to the flux approximation

$$f = -((\Phi_{i+1} - \Phi_i)/h)(2K_i K_{i+1}/(K_i + K_{i+1}))$$

where the coefficient $2K_i K_{i+1}/(K_i + K_{i+1})$ is the harmonic mean ([4]).

The generalization of this approximation is summarized below for triangles, quadrilaterals are discussed in section 1.9. Further details can be found in [2], [6], [7] and [8].

Figure 1.1: Control Volume: Flux Continuity and Cell Face Indices

Continuous interface pressures are introduced at a, b, c as indicated in Fig. 1.1, and normal flux continuity is imposed across the control volume interfaces at the continuous pressure locations. The continuity constraints at the interfaces can be expressed as follows:

$$\begin{aligned}
\mathbf{V}_1 \cdot \mathbf{n}_{ad} &= \mathbf{V}_2 \cdot \mathbf{n}_{ad} \\
\mathbf{V}_2 \cdot \mathbf{n}_{bd} &= \mathbf{V}_3 \cdot \mathbf{n}_{bd} \\
\mathbf{V}_3 \cdot \mathbf{n}_{cd} &= \mathbf{V}_1 \cdot \mathbf{n}_{cd}
\end{aligned} \tag{1.3}$$

where $\mathbf{V}_i$ is the controlvolume velocity for ($i = 1, 2, 3$), and e.g. $\mathbf{n}_{ad}$ is the outward normal on the interface a-d. The potential Φ is assumed to have a linear spatial variation over each interior triangle e.g. $(1, a, c)$ and consequently (by Darcy's law Eq. 1.2) the flux is linear in Φ . Thus Eq. 1.3 can be used to express the interface potentials in terms of the adjacent vertex potentials Φ_i for ($i = 1, 2, 3$). The outward normal flux corresponding to a given control volume face can then be expressed as a linear combination of the cell vertex potentials and written as

$$F_{e(q)}^m(\Phi) = \sum_{l=1}^m \alpha_l \Phi_l \tag{1.4}$$

where α_l are the generalized transmissibility coefficients, $e(q)$ is the edge to cell index and summation is over the local vertex number ($m = 4$ for a quad, 3 for a triangle). For example the flux normal to control volume face ad is given by

$$F_{e(1)}^m = - \int_a^d \mathbf{V}_1 \cdot \mathbf{n}_{ad} dl = -A_{ad} \mathbf{V}_1 \cdot \mathbf{n}_{ad}$$

where A_{ad} is the length (surface area in 3-D) of the interface a-d. The net edge based flux for edge $E(j)$, that connects the i and j vertices (Fig. 1.1), is denoted by

$$F_{E(j)}^m(\Phi) = \sum_{q=1}^{nc} F_{e(q)}^m(\Phi) \tag{1.5}$$

where summation is over the control volume faces intersecting the given edge $nc = 1$ at a boundary or 2 otherwise (in 2-D). A fully Implicit formulation [7] is employed to solve Eq. 1.1 and is written in the concise locally conservative integral form

$$\Phi \left(s_i^{n+1} - s_i^n \right) \tau_i + \Delta t \sum_{j=1}^{Ned} \lambda \left(s_{E(j)}^{n+1} \right) F_{E(j)}^m \left(\Phi^{n+1} \right) = M_j \tag{1.6}$$

where τ_i is the actual physical volume of the control volume, M_j is a specified phase flux and summation is over all Ned edges connected to vertex i . The hyperbolic flux contribution is upwinded according to wave direction along each edge, for a positive outward wave with respect to i , $s_{E(j)}^{n+1} = s_i^{n+1}$, otherwise $s_{E(j)}^{n+1} = s_j^{n+1}$.

1.6 Flow in a Channel with High Curvature

Grid flexibility is demonstrated by considering the case of a meandering channel with high permeability that constrains the flow to follow a specific path. Due to the channel curvature it is not possible to accurately represent the channel geometry using a standard uniform Cartesian grid with a finite number of cells. The channel is assigned a high permeability of 1000 md, while the rest of the reservoir has a permeability of 10 md (See Figure 1.2) and the permeability is assumed to be isotropic.

Figure 1.2: Channel Permeability Map

Two approaches to gridding are considered, (a) the Boundary Aligned Grid that involves moving nodes to be aligned with interior boundaries and (b) Modular Refinement which involves refinement in regions containing interior boundaries. These two grid types are compared with standard Cartesian grids used in conventional simulators. The reference solutions are given by ECLIPSE using a 16×16 coarse grid and 128×128 fine grid respectively.

1.6.1 Boundary Aligned Grid (BAG-Grid)

The BAG-Grid begins with a Delaunay triangulation of a set of grid nodes. After triangulation, the control volumes are either defined as Voronoi (joining triangle circumcenters to edge mid-points) or CVFE (joining triangle barycenters to edge mid-points). The initial grid can then be adjusted according to contours of heterogeneities with well defined internal boundaries. This process involves moving the nodes of local control volumes that are adjacent to the boundary contours so that the gridblock faces are aligned with the contours.

However the grid must be chosen such that the boundary line only passes through two sides of a given triangle, otherwise an ambiguity is raised in how to move the nodes. This is illustrated in figure 1.3 below:

Figure 1.3: Limits on Curvature

When the BAG option is employed, the centernodes of triangles adjacent to the internal boundary are placed on the boundary such that control volume interfaces are boundary-aligned. In the case of a Voronoi grid, such an approach violates the Voronoi property due to the loss of local orthogonality. Alternative approaches involve either using a CVFE grid globally throughout the domain, or locally where gridblocks are distorted to fit the internal boundary while retaining the Voronoi structure elsewhere in the domain. The CVFE-BAG grid of control-volumes corresponding to the channel presented above is shown in Fig 1.4. The boundaries of this channel have been defined as contours and control-volume faces have been aligned locally with the heterogeneity contours.

Figure 1.4: Boundary Aligned Grid for Channel

1.6.2 Local Modular Refinement

In this method, base grid nodes are defined by a coarse Cartesian mesh and locally refined Cartesian grid modules are placed over the channel. The resulting field of grid nodes is then triangulated and finally control volumes are defined by joining triangle barycenters with cell edge mid-points. The location and size of the modules must be chosen such that the resulting point distribution leads to a triangulation with minimum distortion (i.e. avoiding triangles with small angles).

Figure 1.5 shows the resulting CVFE grid built by covering the channel with Cartesian grid modules followed by triangulation.

Figure 1.5: Modular Refinement CVFE Grid for Channel

1.6.3 Results

The data used in all test cases is presented in the Appendix. Two phase flow (oil-water) is simulated. Boundary conditions are defined by an injector and a producer, that are completed along the left and right hand boundaries of the reservoir respectively.

The performance of the above unstructured grid methods is compared with conventional coarse and fine Cartesian grid simulations. Performance is determined by comparing predicted water-cuts. Water-cut is a sensitive simulation parameter and is used to determine precise time of breakthrough. Computed water-cuts are presented in Fig. 1.6.

Modular Refinement: The water-cut response computed with local modular refinement shows a significant improvement over the coarse 16x16 Cartesian grid result. With only a modest increase in the net number of cells used, 469 in this case, that are mainly

Figure 1.6: WaterCut Comparison: BAG Grid (top) and Local Modular Refinement (bottom)

concentrated in the channel, the grid provides improved resolution of the channel boundary and the ensuing flow field.

BAG: The boundary aligned grid significantly improves resolution without increasing the number of gridblocks even when compared to the locally refined grid result. This result clearly demonstrates the importance of aligning the grid with flow boundaries.

The only limitation of both of these methods is in the dependence on known internal boundary locations, which requires an accurate description of reservoir geometry. While modular refinement does not depend on such a precise definition of critical flow boundaries, maximum efficiency will only occur with minimum sized modules and therefore boundary detection is still important.

In conclusion both the BAG and modular refinement methods, in the form described above, can be effective provided that interior boundaries are known *a priori*.

1.7 Streamline Grids and Discretization

Streamline methods have gained revitalized interest in reservoir simulation due to the recent developments of [[9] and [10] and [11]], which show that the streamline method is a very efficient way to model displacement processes in general heterogeneous reservoirs.

Inspired by this success, a first attempt at streamline grid generation [1] involving grids composed of streamlines and constant X-lines demonstrated a significant improvement in water-cut response when compared with a uniform Cartesian grid using the same number of grid nodes.

In this work a natural streamline-equipotential coordinate system is developed for generating grids in heterogeneous media. The inherent approximate-orthogonality in these grids minimizes grid skewness which can occur with the above approach. The method is coupled with FLEX and is within the same object oriented framework. The resulting grids reflect variations in the pressure field caused by permeability heterogeneities in the reservoir and can cluster grid lines in high flow regions, which in turn can be expected to increase local accuracy of the finite-volume discretization.

In contrast to the methods presented in the previous section which require that flow boundaries be known *a priori*, here by definition, the streamlines are aligned with flow boundaries which are determined within the grid generation procedure.

The grid generation process involves: (a) a fine scale Cartesian grid single phase flow calculation using the streamline method [9] to trace streamlines calculated from the fine grid velocity field (b) selection of desired streamlines to a specified coarse grid resolution,

(c) projection of the pressure field via interpolation onto the selected streamlines, followed by isopotential line interpolation across the selected streamlines, and (d) upscaling of fine grid permeabilities onto the new coarse grid.

Currently power law upscaling is used to define the coarse curvilinear grid permeability field from fine scale rock data, while rock curves are used directly to define the relative permeabilities on the coarse streamline grid. The grid vertices are defined by the intersections between the streamlines and isopotential lines. Away from grid singularities the grid is generally comprised of quadrilateral cells.

1.8 General Curvilinear Coordinates

First we focus on the single phase pressure equation. The problem is to find the pressure Φ satisfying

$$\int_{\Omega} -\nabla \cdot \mathbf{K}(x, y) \nabla \Phi d\tau = M \quad (1.7)$$

over an arbitrary domain Ω , subject to Neumann/Dirichlet conditions on boundary $\partial\Omega$. The right handside term M represents a flow rate, and $\nabla = (\partial_x, \partial_y)$. As before Matrix $\mathbf{K}$ can be a diagonal or full Cartesian tensor with general form

$$\mathbf{K} = \begin{pmatrix} K_{\alpha\alpha} & K_{\alpha\beta} \\ K_{\alpha\beta} & K_{\beta\beta} \end{pmatrix} \quad (1.8)$$

and is elliptic such that

$$K_{\alpha\beta}^2 \leq K_{\alpha\alpha} K_{\beta\beta} \quad (1.9)$$

The pressure equation is defined above with respect to the physical Cartesian tensor in the initial classical Cartesian coordinate system. Equation 1.7 is reformulated with respect to a curvilinear coordinate system that has a precise correspondence with a uniform dimensionless logically rectangular Cartesian (ξ, η) coordinate system. Performing the integration with respect to an arbitrary control volume Ω_p comprised of surfaces that are tangential to constant (ξ, η) respectively equation 1.7 can be expressed as

$$-\int_{\Omega} \nabla \cdot \mathbf{K}(x, y) \nabla \Phi d\tau = -\int \int_{\Omega_p} (\partial_{\xi} \tilde{F} + \partial_{\eta} \tilde{G}) d\xi d\eta = \Delta_{\xi} F + \Delta_{\eta} G = M \quad (1.10)$$

where F and G are the general tensor flux components.

$$\begin{aligned}\tilde{F} &= - \int (T_{aa}\Phi_\xi + T_{ab}\Phi_\eta) d\eta \\ \tilde{G} &= - \int (T_{ba}\Phi_\xi + T_{bb}\Phi_\eta) d\xi\end{aligned}\tag{1.11}$$

$\tilde{F}$ and $\tilde{G}$ are the integrands of Eq. 1.1 and $\Delta_\xi F$ is the difference in net flux with respect to ξ . General tensors are defined by

$$\begin{aligned}T_{aa} &= (K_{\alpha\alpha}y_\eta^2 + K_{\beta\beta}x_\eta^2 - 2K_{\alpha\beta}x_\eta y_\eta)/J \\ T_{ab} &= (K_{\alpha\beta}(x_\xi y_\eta + x_\eta y_\xi) - (K_{\alpha\alpha}y_\eta y_\xi + K_{\beta\beta}x_\xi x_\eta))/J \\ T_{bb} &= (K_{\alpha\alpha}y_\xi^2 + K_{\beta\beta}x_\xi^2 - 2K_{\alpha\beta}x_\xi y_\xi)/J\end{aligned}\tag{1.12}$$

where $J = x_\xi y_\eta - x_\eta y_\xi$. Off diagonal tensor coefficients play a critical role in the presence of cross bedding, anisotropy, upscaling and non-Cartesian grids. Further details of this derivation are presented in [12] where it is concluded that omission of the cross term T_{ab} can lead to a significant $O(1)$ error in flux. Also any scheme applicable to a full tensor can be applied to a general curvilinear grid by defining the tensor coefficients through equations 1.12.

1.9 Quadrilateral Discretization

Three variants of a family of flux continuous finite volume schemes are considered. The first two are cell centered where flow variables and rock properties are defined at the grid cell centers and the third is vertex centered or point distributed where flow variables and rock properties are defined at the grid vertices.

1.9.1 Numerical Scheme

The finite volume schemes employed here are based on [6] who present a framework for locally conservative flux continuous discretization of the full tensor pressure equation. A similar scheme is also presented by [7], and the triangulation described in section 1.2 follows a similar approach. Flow variables and rock properties are assumed to be defined at the nodes of a generic quadrilateral mesh as indicated in Fig 1.7 where nodes are numbered 1 to 4. It is important to notice that the nodes can either be at grid cell centers or cell vertices and thus the formulation can either be cell centered or cell vertex/point distributed. Flux and pressure continuity are achieved by introducing auxiliary cell interface pressures at a, b, c and d Figure 1.7 :

Figure 1.7: Points of Continuity in Flux and Pressure

The potential Φ assumes a linear variation over each of the shaded triangles illustrated in Figure 1.7. Using the definition of flux given by Eq. 1.11, flux continuity is imposed at the pressure continuity points a, b, c and d of Fig. 1.7 and result in four equations for the four additional interface pressures introduced into the system, where

$$\begin{aligned}
 (T_{aa}\Phi_\xi + T_{ab}\Phi_\eta)|_a^1 &= (T_{aa}\Phi_\xi + T_{ab}\Phi_\eta)|_a^2 \\
 (T_{ab}\Phi_\xi + T_{bb}\Phi_\eta)|_b^2 &= (T_{ab}\Phi_\xi + T_{bb}\Phi_\eta)|_b^3 \\
 (T_{aa}\Phi_\xi + T_{ab}\Phi_\eta)|_c^3 &= (T_{aa}\Phi_\xi + T_{ab}\Phi_\eta)|_c^4 \\
 (T_{ab}\Phi_\xi + T_{bb}\Phi_\eta)|_d^4 &= (T_{ab}\Phi_\xi + T_{bb}\Phi_\eta)|_d^1
 \end{aligned} \tag{1.13}$$

The system of Eq 1.13 can be rearranged so as to express the interface pressures as a linear combination of the four adjacent pressures at the nodes (1, 2, 3, 4) Fig. 1.7, and thus the interface pressures can be eliminated from the system. In general the outward normal flux can be expressed as a linear combination of the m node pressures with $m = 4$ for a quadrilateral grid cell ($m = 3$ for a triangle) c.f. Eq. 1.4. By performing this operation for each group of four cells in a pre-processing step, and assembling cell face fluxes two per node (e.g. node 1 receives fluxes from a and d), a family of locally conservative flux continuous nine-point schemes is derived, further details are given in [6]. The implicit scheme can then be assembled as before, by summing fluxes adjacent to the cell edges passing through the given *ith* vertex c.f. Eqs. 1.5 and 1.6. The above procedure can be regarded as a full tensor generalization of the derivation of the harmonic mean. If only two nodes are involved the procedure reduces to a two point flux with coefficient defined by the harmonic mean. Three variants of this formulation are described below.

1.9.2 (a) Cell-centered: Transform Space Flux Continuity

In this approach [13] the grid in the physical domain is mapped to a uniform Cartesian grid in the computational domain in a cell-wise fashion. Each quadrilateral cell is

mapped to a unit Cartesian cell through the isoparametric transformation

$$\mathbf{r} = (1 - \xi)(1 - \eta)\mathbf{r}_{i,j} + \xi(1 - \eta)\mathbf{r}_{i+1,j} + \xi\eta\mathbf{r}_{i+1,j+1} + (1 - \xi)\eta\mathbf{r}_{i,j+1} \quad (1.14)$$

Tensor transform derivatives are found by differentiating Eq. 1.14. Thus a physical grid cell with Cartesian permeability tensor is mapped to a square grid cell with a general tensor given by 1.12 (see Figure 1.8). A piece-wise constant general tensor can be defined for each cell by evaluating transform derivatives at the cell centers. If the cell-wise constant Cartesian tensor is symmetric, then the general tensor is also cellwise constant and symmetric. For example referring to 1.13

$$T_{ab}|_a^1 = T_{ab}|_d^1 \quad (1.15)$$

This property is necessary for constructing a scheme with a symmetric positive definite matrix [8]. This approach also simplifies the definition of the general tensor in terms of geometry, and simplifies implementation in two and three dimensions since all aspects of geometry can be defined through cell-wise piecewise constant tensors. However, in this case even for boundary aligned grids, since approximate cell face data is included in the flux approximation, some smoothing of boundary data is incurred. In effect the exact boundary location is approximated to second order, convergence of the scheme is studied in [12].

Figure 1.8: Transformation: Physical Cell - Computational Cell

1.9.3 (b) Cell-centered: Physical Space Flux Continuity

If the constraints of 1.13 are forced directly on the physical grid, then by Eq. 1.11 the general tensors are functions of transform derivatives evaluated directly at the cell edge. Therefore for an arbitrary quadrilateral, the edge based tensors will not generally be symmetric with respect to the cell and in contrast to 1.15

$$T_{ab}|_a^1 \neq T_{ab}|_d^1 \quad (1.16)$$

due to the difference in cell edge geometry at edges a and d. Consequently the discrete matrix cannot be shown or expected to be symmetric [12]. Physical space continuity also

involves a more complicated implementation (in 2 and 3-D) since each flux must be defined with respect to the local cell face geometry, the grid cell being the control volume. However, if grid cells are geometrically aligned with boundaries, then the boundaries are represented with precision since the exact cell-edge geometry is built into the flux approximation. The effects of geometry discretization are discussed further in the results section.

1.9.4 (c) Point Distributed: Physical Space Flux Continuity

In this case flow variables and rock properties are defined at the grid vertices where the streamlines and isopotential lines intersect and the discretization is essentially the dual of the cell centered scheme (b) described above. For a given grid vertex the control volume in this method is defined by joining the barycenters of the surrounding grid cells with the midpoints of the cell edges that pass through the given vertex, Fig. 1.9. A maximum of 8 faces is allowed for each control volume (in 2-D), which ensures that the grid respects the curvatures of reservoir heterogeneities and channels. Continuity conditions are enforced in physical space across the control volume faces and will again result in a non-symmetric tensor, while geometry is precisely represented. Finally we note that while cell centered schemes are directly compatible with conventional upscaling methods, for an irregular grid of given level, it is well known that the point distributed (cell vertex based) scheme is more accurate than a cell centered scheme in terms of truncation error [4].

1.10 Results

The performance of the above schemes using streamline-potential grids is measured against conventional coarse and fine Cartesian grid simulations for three test cases described below. Each case involves simulation of two phase flow. Method performance is judged primarily from a comparison of predicted water-cuts. Water-cut is a sensitive simulation parameter and is used to determine precise time of breakthrough. Fine scale Cartesian grid results are used to provide reference solutions and the calculation of streamlines is performed by solving the single phase flow equation on the same fine scale grid. A coarse set of streamlines is then selected and the procedure described above is used to determine the final streamline-equipotential grid and permeability field.

■ Point Distributed Control Volume

Figure 1.9: Point Distributed Scheme: Grid and ControlVolume

■ Cell Centered Control Volume

Figure 1.10: Cell Centered Scheme : Grid and ControlVolume

1.10.1 Heterogeneous Field

The first test case consists of a highly heterogeneous permeability field with a 30% correlation length in the horizontal direction Fig. 1.11. In each case completed injector and producing wells are imposed on the left and right hand side boundaries. The streamline-potential grid is shown in Fig. 1.12. A comparison of water-cuts predicted by the respective ECLIPSE fine grid, and coarse grid simulations and FLEX using scheme (a) with the streamline-potential grid are shown in Fig. 1.13. A notable improvement in water-cut is indicated by the new method, (in particular breakthrough prediction) when compared with coarse grid ECLIPSE. The streamline-potential grid resolution is 26×13 compared to the 256×128 Cartesian fine grid. A comparison with the other schemes described in section 1.8 above is shown in Fig 1.14, in this case all three schemes predict a similar improvement in water-cut response. Oil recoveries are shown in Fig. 1.13 for scheme (a) and serve to emphasize the improved performance due to the streamline-potential grid.

Figure 1.11: Heterogeneous Permeability Field

Figure 1.12: Streamline Grid with Heterogeneous Flow Paths

Figure 1.13: Water-Cut and Cumulative Oil Production

Figure 1.14: Water-Cut: Comparison between schemes a, b, c

1.10.2 Meandering Homogeneous Channel

In the second example we return to the meandering channel of section 1.6 c.f. Fig 1.2. The channel is assigned a constant, isotropic, homogeneous permeability that is two orders of magnitude larger than the rest of the otherwise constant field

The streamline-potential grid is presented in Fig. 1.15. while the grid is highly distorted, it is observed that the streamlines are naturally aligned with the channel over most of the reservoir. Again this illustrates the natural ability of the method to align the grid with crucial heterogeneities, and is in contrast with the methods of section 1.6, which both require *a priori* knowledge of the flow channel before gridding can take place. As in the previous case, channel flow boundary conditions with completed wells either side of the domain are imposed and are used for both generating the grid and testing performance.

The watercut comparison between the Cartesian reference solutions and that of scheme (a) using the streamline-potential grid is shown in Fig. 1.16. A dramatic improvement in prediction of this key parameter is observed. In contrast to the ECLIPSE 16x16 coarse grid, the FLEX streamline-potential grid solution indicates an improvement in the estimate of breakthrough by approximately 40%, clearly demonstrating the benefit of employing the streamline-potential grid.

Scheme (b) provides even further improvement in resolution compared to scheme (a) which can be seen in Fig. 1.17. The difference in results computed by schemes (a) and (b) can be attributed to the difference in formulation in the types of scheme. As discussed in section 1.9 scheme (a) involves some smoothing of the grid transform derivatives and thus geometry effects, while scheme (b) maintains a precise definition of grid geometry in the flux at the expense of a more involved implementation and introduction of a non-symmetric tensor. Scheme (c) (which is point distributed) also maintains a precise definition of grid geometry and consequently also has a more involved implementation, however results are comparable with that of scheme (b).

Finally we note that cell centered scheme grid blocks are precisely the grid cells c.f. Fig. 1.15, while the point distributed scheme grid blocks are polygons c.f. Fig. 1.18.

1.10.3 Meandering Heterogeneous Channel

The last example involves an anisotropic heterogeneous version of the channel problem, where a varying permeability field is imposed within the channel and is shown in Fig. 1.19. The anisotropy ratio is given by $K_y/K_x = 10$. Again the streamlines are naturally clustered in high permeability regions and the streamline-potential grid is shown in Fig. 1.20.

Figure 1.15: Streamline Grid for a Homogeneous Isotropic Channel

The observed trends in scheme performance are similar to the previous case. The water-cut computed by scheme (a) using the streamline-potential grid shows a significant improvement when compared with the coarse Cartesian grid result Fig. 1.21. Further improvement is obtained by using schemes (b) and (c), and as before the additional improvement is attributed to enforcing the flux continuity constraint directly on the physical grid, Fig. 1.22. Finally a comparison between the evolution of the saturation field computed by ECLIPSE 16x16 and that by scheme (c) using the streamline-potential grid is shown in Figs 1.23 and 1.24. The saturation maps clearly demonstrate the reduction in numerical diffusion due to the comparatively locally fine grid that is aligned with critical flow regions, in particular the early breakthrough predicted by the streamline-potential grid computation is almost apparent in results after 20 days. A separate body of evidence demonstrating streamline and streamtube benefits is presented in [14].

Figure 1.16: WATER CUT (Homogeneous Isotropic Channel)

Figure 1.17: Water-Cut Homogeneous Channel: Schemes a, b, c

Figure 1.18: Generalized Point Distributed Grid (Homogeneous/Isotropic Channel)

Figure 1.19: Heterogeneous Channel Permeability Distribution

Figure 1.20: Heterogeneous Channel Streamline Grid

Figure 1.21: WATER CUT : Heterogeneous Channel

Figure 1.22: Water-Cut Heterogeneous Channel: Schemes a, b, c

Figure 1.23: Streamline Grid Saturation Profiles (Scheme c)

Figure 1.24: ECLIPSE Saturation Profiles

1.11 Conclusions and Future Work

A new grid generation technique based on streamlines and equipotentials is developed for reservoir simulation. The method relies on the natural property of streamlines to provide a grid that has a concentration of grid lines in high velocity regions of the flow field thereby naturally optimizing nodal placement. The method is implemented in C++ and is coupled with FLEX. The streamline tracing is performed using the 3DSL streamline code [10] developed at Stanford.

The use of a full tensor scheme ensures that a consistent approximation is retained for any grid type. Both point-distributed and cell centered variants of the flux continuous scheme are further developed and contrasted for highly distorted grids. While local mapping of the physical domain to the computational domain conveniently builds the geometry into the tensor, calculation of continuous fluxes directly on the physical grid improves solution quality.

Results computed using the new grids demonstrate a significant improvement in accuracy when compared with conventional Cartesian grids of similar resolution. Streamline grids and unstructured grids are shown to be particularly effective when aligned with dominant channel boundaries. The streamline grid has the added advantages of natural boundary alignment with crucial flow paths while concentrating streamlines within the dominant flow paths of the reservoir.

In addition it is observed that streamline-potential grids lead to more accurate solutions in terms of numerical diffusion and grid orientation effects. In particular, comparisons of critical breakthrough parameters show that results are very close to fine Cartesian grid results, while the method drastically reduces the number of gridblocks used.

Currently, streamlines are generated for specific well configurations and reservoir shape. Future development will involve generalization of the method to arbitrary well configurations. We anticipate the need for a mixed element grid involving both triangles and quadrilaterals in the case of natural grid singularities, e.g. at isolated wells and regions where there is a loss of logically rectangular connectivity. Work is now beginning on the development of streamline grid generation in a modular framework. Looking further ahead, we

plan to consider dynamic grid generation which would have a variety of applications including treatment of changes in boundary conditions such as the introduction or closing of a well.

In the shorter term, further development of the streamline tracing algorithm is required in order to treat full tensor velocity fields on structured and unstructured grids. The FLEX code can also be exploited for use in this preprocessing step.

No attempt has been made in this work to study the crucial problem of upscaling. Fine scale permeability distributions are currently upscaled to control-volume polygons by power law averaging. There is a need for development of a robust full tensor upscaling algorithm that is applicable to general polygons in two and three dimensions.

The work presented here is in two dimensions. The generation and development of general three-dimensional streamline grids is a requirement that would form the basis of a major project. The interaction and effects of gravity on streamline grids will also require further study.

Nomenclature

$d\tau$ = volume increment

$e(q)$ = edge to cell suffix

F, G = normal cell face flux components L^2/T in 2-D

K = permeability tensor L^2

J = Jacobian of a cell L^2

k_{rj} = relative permeability of j^{th} phase dimensionless

M_j = j^{th} phase flux L^2/T

n = time level superfix

r = position vector L

s_j = saturation of j^{th} phase - dimensionless

t = time T

T = general geometry-permeability tensor L^2

v_j = velocity of j^{th} phase L/T

V_l = discrete velocity at node l L/T

x, y = Cartesian coordinates L

α_i = discrete generalized transmissibility coefficient
 λ_j = mobility of j^{th} phase L^3T/M
 μ_j = viscosity of j^{th} phase M/TL
 ϕ = porosity dimensionless
 Φ = potential M/T^2L
 ξ, η = uniform dimensionless transform-space coordinates

References

- [1] S. Verma. *Flexible Grids for Reservoir Simulation*. PhD thesis, Stanford University, 1996.
- [2] S. Verma and K. Aziz. Two- and three-dimensional flexible grids in reservoir simulation. In *5th European Conference on the Mathematics of Oil Recovery*, Loeben, Austria, 3-6 Sept 1996.
- [3] C.L. Palagi. *Generation and Application of Voronoi Grids to Model Flow in Heterogeneous Reservoirs*. PhD thesis, Stanford University, May 1992.
- [4] K. Aziz and A. Settari. *Petroleum Reservoir Simulation*. Applied Science Publishers Ltd. (London), 1979.
- [5] K. Aziz. *Notes for Petroleum Reservoir Simulation*. Stanford University (Stanford), 1995.
- [6] M.G. Edwards and C.F. Rogers. A flux continuous scheme for the full tensor pressure equation. In *4th European Conference on the Mathematics of Oil Recovery*, Norway, 1994.
- [7] O. Boe I. Aavatsmark, T. Barkve and T. Mannseth. Discretization on nonorthogonal curvilinear grids for multiphase flow. In *4th European Conference on the Mathematics of Oil Recovery*, Roros, 1994.
- [8] T. Barkve I. Aavatsmark and T. Mannseth. Control-volume discretization methods for 3d quadrilateral grids in inhomogeneous and anisotropic reservoirs. In *SPE Reservoir Simulation Symposium*, Dallas TX USA, 1997.

- [9] M.R. Thiele, R.P. Batycky, M.J. Blunt, and F.M. Orr. Simulating flow in heterogeneous systems using streamtubes and streamlines. *SPERE*, 11(1):5-12, 1996.
- [10] R.P. Batycky. *Field Scale 3-D Streamline Simulator - User Manual*. Stanford University (Stanford), 1997.
- [11] T. Gimse F. Bratvedt and C. Tegnander. Streamline computations for porous media flow including gravity. *Transport in Porous Media*, 25(1):63-78, 1996.
- [12] M.G. Edwards and C.F. Rogers. A finite volume scheme with imposed flux continuity for the full tensor pressure equation. *J Comput Geosci (Submitted)*, 1995.
- [13] M.G. Edwards. Superconvergent renormalization and tensor approximation. In *5th European Conference on the Mathematics of Oil Recovery*, Loeben, Austria, 3-6 September 1996.
- [14] R. C. M. Portella and T. A. Hewett. Gridding, upscaling and simulation using stream-tube techniques. *SCRF 11th Annual Report, Stanford University*, 11, 1998.

Appendix - Data Used in the Test Examples

Item	Field Units	SI Units
Reference Pressure	14.7 <i>psia</i>	$1.0128 \times 10^5 \text{ Pa}$
Reference Temperature	60 <i>degF</i>	288.71 <i>Kelvin</i>
Oil Properties		
Density at reference conditions	49.942 <i>lbm/ft</i> ³	800.00 <i>kg/m</i> ³
Formation Volume Factor	1.03 <i>RB/STB</i>	1.03 <i>Rm</i> ³ / <i>Sm</i> ³
Compressibility	$1.9 \times 10^{-5} \text{ psi}^{-1}$	$2.7557 \times 10^{-9} \text{ Pa}^{-1}$
Viscosity at reference conditions	1.1 <i>cp</i>	$1.1 \times 10^{-3} \text{ Pa.s}$
Water Properties		
Density at reference conditions	62.3664 <i>lbm/ft</i> ³	999.973 <i>kg/m</i> ³
Formation Volume Factor	1.02 <i>RB/STB</i>	1.02 <i>Rm</i> ³ / <i>Sm</i> ³
Compressibility	$2.5 \times 10^{-5} \text{ psi}^{-1}$	$3.6259 \times 10^{-9} \text{ Pa}^{-1}$
Viscosity at reference conditions	0.55 <i>cp</i>	$0.55 \times 10^{-3} \text{ Pa.s}$
Rock Properties		
Compressibility	$3.1 \times 10^{-6} \text{ psi}^{-1}$	$4.4962 \times 10^{-10} \text{ Pa}^{-1}$
Porosity at reference conditions	0.25	0.25
Capillary pressure	0 <i>psi</i>	0 <i>Pa</i>

Table 1.1: Reservoir and Rock Data for Test Problem

SW	KRW	KRO
0.0	0.0	1.0
0.1	0.0	0.96
0.2	0.0	0.875
0.25	0.0	0.8
0.3	0.01	0.7
0.4	0.04	0.43
0.5	0.075	0.23
0.6	0.125	0.1
0.7	0.20	0.025
0.8	0.30	0.0
0.9	0.575	0.0
1.0	1.0	0.0

Table 1.2: Relative Permeability Table

2. Well Index Correlation for a Finite Conductive Horizontal Well

by Chris Wolfsteiner, Sepehr Arbabi, and Khalid Aziz

Abstract

This work investigates the productivity losses due to frictional and accelerational pressure drops in a horizontal well. Comparison between a commercial numerical simulator and an semi-analytical solution is made on basis of well indices. The results are used to determine correct well indices. Several cases show how the results of a numerical simulator can be improved hereby.

2.1 Introduction

Horizontal wells provide both opportunities and challenges in reservoir development. In our evaluation and prediction of reservoir performance the use of numerical flow simulators is a necessity.

The representation of wells in a simulator is a rather critical task. For short perforation intervals the infinite conductive (constant pressure) well model is sufficiently accurate. When considering wellbore pressure drops, the contribution of a hydrostatic pressure difference is most important for vertical wells. Other sources causing resistance to the flow within the production pipe are usually neglected, to avoid rather expensive computations.

The modeling of horizontal wells evolved in analogy to vertical wells. But horizontal wells are very different. First, in absence of a hydrostatic pressure drop, frictional and accelerational affects may have an important impact on the well productivity. Second, normal perforation lengths of several thousand feet combined with large flow rates make these considerations even more important.

Horizontal wells are often modeled incorrectly in numerical reservoir simulators. Well block pressures are related to bottom hole flowing pressures using analytical models under severe simplifying assumptions. Correlations for friction losses are available optionally but inflow (accelerational) effects are still neglected.

2.2 Conventional Well Model

The standard well model used in most numerical simulators was introduced by Peaceman [[1], [2]]. It operates under the following assumptions:

- single phase fluid
- (pseudo) steady state flow
- strictly radial (two dimensional) flow
- no compressibility
- well is close to the center of the reservoir

The basic equations (well model) relating the well pressure p^w to the well block pressure p_0 at a given well flow rate q^w are

$$WI = \frac{2\pi kh}{\ln \frac{r_0}{r_w} + s} \quad (2.1)$$

$$p^w = p_0 + \frac{q^w}{T^w} = p_0 + \frac{q^w}{WI \cdot \lambda} \quad (2.2)$$

The *well transmissibility* T^w is defined as the *well index* WI times the mobility $\lambda_p = \frac{k_r p}{\mu B}$. This model is derived under the the assumption of single phase flow, it is also used for multiphase problems by adding λ_p in the absence of a better theory. Using Eqs. 2.1 and 2.2, r_0 can be expressed as a function of p_0 , p^w and q^w for a given well segment.

$$r_0 = r_w \exp \left[2\pi kh \frac{(p^w - p_0)}{q^w} \lambda - s \right] \quad (2.3)$$

The *equivalent wellblock radius* r_0 is the distance from the block node at which the analytical model in Eq. 2.2 gives the same pressure as the well block pressure. It depends only on the geometry of the block containing the well and permeabilities. For a Cartesian grid with block spacing $\Delta x, \Delta y$ and Δz in an anisotropic reservoir with permeabilities k_x, k_y and k_z and a vertical well it is

$$r_0 = 0.28 \frac{\sqrt{\sqrt{\frac{k_y}{k_x}} \Delta x^2 + \sqrt{\frac{k_x}{k_y}} \Delta y^2}}{\sqrt[4]{\frac{k_y}{k_x}} + \sqrt[4]{\frac{k_x}{k_y}}} \quad (2.4)$$

More general geometries have been considered in literature. Under isotropic conditions ($k_x = k_y = k_z$) Eq. 2.4 reduces to

$$r_0 = 0.140365\sqrt{\Delta x^2 + \Delta y^2} \quad (2.5)$$

and hence a *normalized equivalent wellblock radius* may be defined as

$$r_{0n} = \frac{r_0}{\sqrt{\Delta x^2 + \Delta y^2}} \quad (2.6)$$

For a grid aspect ratio $\alpha = \frac{\Delta y}{\Delta x}$ of unity, we have the classical result of $r_{0n} = 0.195$.

2.3 Semi-Analytical Model

An analytical model was introduced by Babu and Odeh [3]. It is a solution of the diffusivity equation for a line source in a box shaped reservoir. The well is aligned in parallel to any edge of the reservoir. Boundary conditions for the reservoir are no-flow and uniform flux for the wellbore. The more appropriate condition of infinite conductivity (constant pressure) for the well is avoided for mathematical reasons. The fluid is considered to be single phase and slightly compressible. The solution covers both transient and pseudo-steady state flow regimes. Permeability anisotropy is also accounted for by scaling the original reservoir to an equivalent isotropic one. The cross-section of the well is not circular anymore in the transformed space.

Penmatcha [5] uses this solution to compute the performance of wells in a box shaped reservoir. A transient, three-dimensional and semi-analytical model for a coupled reservoir / wellbore system was obtained. The well was split into equal length segments and solutions for these segments were superimposed in space and time. Due to the segmentation, an infinite conductivity condition can be achieved. The model produces a flow rate and a well pressure for each segment and timestep. The program is also capable of resembling the finite conductivity well i.e. when wellbore pressure drop is present. In this case, a momentum equation is included for pressure drop between segments which accounts for frictional and accelerational (inflow) pressure drop as described in more detail below.

Figure 2.1: f vs. Re ; Four Approximations for the Fanning Friction Factor (fully turbulent, $\epsilon = 0.0005$)

2.4 Well Pressure Drop

Whenever flow in pipes is part of a problem we are interested in the amount of work to move the fluid. The classical treatment of this issue considers the pressure loss due to hydrostatic head and pipe roughness.

$$\Delta p = \Delta p_{head} + \Delta p_{fric} = \rho g h + f \frac{L}{D} \rho v^2 \quad (2.7)$$

The contribution from the first term is negligible in this study of horizontal wells. For Δp_{fric} the lack of a strict theory in fluid dynamics requires the use of correlations based on experiments instead: the *Fanning friction factor* f is a function of the dimensionless *Reynolds number* $Re = \frac{D v \rho}{\mu}$ and the *relative pipe roughness* $\epsilon = \frac{k}{D}$. It is obtained from the Moody (1944) friction factor chart. For laminar flow ($Re < 2100$), $f = \frac{16}{Re}$. In case of turbulent flow ($Re > 4000$) f is calculated by the implicit Colebrook [6] equation (Eq. ??) or one of its explicit approximation [7], [8], citechen.

In the transition from laminar to turbulent flow, the friction factor is interpolated between the values at $Re = 2100$ and $Re = 4000$.

In case of wells the theory for closed, rough pipes applies only partially. The fluid entering the perforations radially has impact on the effective friction factor and causes additional pressure drops due to accelerational effects. As shown by Ouyang *et. al* [11], the conventionally obtained friction factor f_0 has to be corrected, for laminar flow

$$f = f_0 \left(1 + 0.04304 Re_w^{0.6142}\right) \quad (2.8)$$

and for turbulent flow

$$f = f_0 \left(1 - 0.0153 Re_w^{0.3978}\right) \quad (2.9)$$

where $Re_w = \frac{qs\rho}{\pi\mu}$ is the *inflow Reynolds number*. The accelerational pressure drop for a well-segment is

$$\Delta p_{acc} = \frac{F_{acc}}{A} = \frac{\rho}{A^2} \left(q_{rad,in}^2 + 2q_{ax,in}q_{rad,in}\right) \quad (2.10)$$

It is a function of axial and radial inflows ($q_{ax,out} = q_{ax,in} + q_{rad,in}$). Both, Eqs. 2.8 resp. 2.9 and 2.10, are considered in the semi-analytical solution.

2.5 Comparative Simulation Study

In this section the results of the semi-analytical solution with those obtained from a commercial reservoir simulator (Eclipse, 1996A) are compared. Deviations of the standard well model from the correct one will be analyzed, a method for early time correction will be shown. Some preliminary comments have to be given first.

Without loss of generality a single horizontal well was aligned in parallel to the y -axis in a block shaped reservoir. Geometry, fluid and reservoir data are given in Table 2.1 and Figure 2.2. In different cases the effect of friction and other parameters was studied. The semi-analytical solution was considered to give the correct results (under the assumptions given). Correct parameters refer to these results in the further discussion. The grid of the simulator has $51 \times 100 \times 5$ blocks in x, y and y -direction. This high resolution ensures that grid effects are kept low.

reservoir length a	6,000	ft
reservoir width b	12,000	ft
reservoir height h	50	ft
permeability $k_x = k_y = k_z$	3,000	mD
porosity ϕ	0.3	
initial reservoir pressure p_{ini}	4000	psi
total compressibility c_t	$3 \cdot 10^{-5}$	psi^{-1}
form. volume factor B_o	1.05	$\frac{RB}{STB}$
viscosity μ	1	cp
density at res. cond. ρ	60	$\frac{\text{lb}_m}{\text{ft}^3}$
max. well rate q_{max}	10,000	$\frac{STB}{d}$
min. bottom hole pressure p_{min}^w	1,200	psi
well location	varying	
well radius r_w	0.1667	ft
skin s	0	

Table 2.1: Data of Basic Reservoir-Well Model

Figure 2.2: Geometry of Wells in a Block Shaped Reservoir.

case	center	offcenter	edge
x_0	3000	1440	120
z_0	25	25	25
D_{ref}	4025	4025	4025
ϵ	0.0005	0.0005	0.0005
reservoir dimensions: 6000 x 12000 x 50 ft			
LONG WELLS: $y_0=3000$ to $y_0=9000$ (50 segs)			
short wells: $y_0=5400$ to $y_0=6600$ (10 segs)			

Table 2.2: Name Conventions for Cases under Study (length unit: feet)

2.5.1 Constraints of the Analytical Model

In order to be able to compare the results of different methods we first need to make sure that we are addressing the same problem with each of them. The problem design is restricted by the less flexible method, which is the analytical one. The following limitations are recognized:

- single phase fluid
- strictly horizontal wells
- constant fluid properties

The problem for the numerical simulator has to be formulated accordingly. The first two requirements are met easily, the third one is trivially fulfilled for horizontal wells.

Fluid properties are commonly interpolated from tables for the simulator. Setting the viscosity μ constant in a table is permissible whereas the formation volume factor B_o has to decrease monotonically with pressure.

The total compressibility for a rock-fluid system (with π phases),

$$c_t = c_\phi + \sum_{\pi} S_{\pi} c_{\pi} \quad (2.11)$$

is frequently encountered in analytical solutions and has no direct analog in the simulator. The slope of the $B_o(p)$ defines already a value for c_o in Eq. 2.11 implicitly ($S_o = 1$ for single phase). It follows that c_ϕ has to be adjusted accordingly to result in a constant value for c_t .

2.5.2 Correct Block Pressure

The semi-analytical solution does not yield block pressures as the reservoir is not discretized. However, for the back calculation of the effective wellblock radii, they are required according to Eq. 2.3. In this approach, values for "correct" p_0 are obtained by prescribing a complete set of analytical flowrates along a well in an equivalent problem for the numerical simulator (Fig. 2.3). This has to be done for every timestep (production schedule). Technically the well has to be split into a set of wells completed in only one block (segments) for the simulator. The block size is chosen to be the same as the well-segment length in the analytical calculation. By this method correct block pressures are obtained. As expected, the prescribed flowrates are exactly reproduced by the simulator. In Fig. 2.3, variables in circles refer to "correct" results, variables in squares to results which are obtained from a standard simulation where wells are not segmented and no values are prescribed. Note also, that the three variables of any simulator output are interrelated by the standard well model i.e. $r_{0n} = 0.14$. The triplet of three variables in circles is used to back calculate the correct well index, as discussed in the following.

2.5.3 Calculation of r_0

Combining the concepts developed in Eqs. 2.3 and 2.6, for a horizontal well in y -direction the correct normalized equivalent wellblock radius is given by

$$r_{0n} = r_w \frac{\exp \left[2\pi k \Delta y \frac{(p^w - p_0) \lambda - s}{q^w} \right]}{\sqrt{\Delta x^2 + \Delta z^2}} \quad (2.12)$$

The numerical simulator uses the basic well model introduced earlier. Applying now Eq. 2.12 with p_0 , p^w and q^w taken from the simulator's output, a value of $r_{0n} = 0.14$ should be reproduced.

The sensitivity of Eq. 2.12 to small deviations from the correct bottom hole flowing pressure is found to be very high. Fig. 2.4 was plotted using data from the numerical simulator run on the "center" case (data in Tabales 2.1 and 2.2). For an arbitrarily chosen segment, $q_w = 522.12 \frac{STB}{d}$, $p_0 = 3991.52$ psi and $p^w = 3990.38$ psi, Eq. 2.12 gives the expected value of 0.14. Since q^w and p_0 are fixed in this study, r_{0n} varies with p^w only. This variable reflects also all the impact of the pressure drop model used.

For a more general discussion, a dimensionless pressure may be defined as (oil field units)

$$p_D^w = \frac{kh}{141.2 q^w B \mu} (p_{ini} - p^w) \quad (2.13)$$

Figure 2.3: A Step Diagram for Computation of Correct Well Index

Figure 2.4: Sensitivity of r_{0n} to p^w for the "center" Case

Both scales are given in Fig. 2.4.

Usually we are interested in p^w rather than r_{0n} . Interpreting Fig. 2.4 in this way, a high tolerance for the wellblock radius can be observed. A deviation of ± 0.2 psi (basic case) translates in a range of 0.055 to 0.35 for r_{0n} ("center" case); for ± 1 psi it is from 0.0013 to 14.7 which covers four orders of magnitude. This implies that extremely high and low values for correct wellblock radii have to be expected because of the exponential relationship in Eq. 2.12.

With this, it appears to be meaningful to carry the concept of the normalized equivalent wellblock radius a little further by representing it as $\ln r_{0n}$ instead. The natural logarithm is proposed because it translates the value of 0.14 in Eq. 2.5 to -1.97 which is close to -2.

2.5.4 Cases: Well Index Analysis and Correction

With these procedures in mind and the data of Table 2.1 and Fig. 2.2, a first case can be presented. We discuss in detail the case where the well is placed near one edge of

the reservoir (case edge in Table 2.2). Plots for some other cases are also given which can be identified by the naming conventions established in Table 2.2. For each case four plots similar to Fig. 2.5 were prepared. The following discussion is general and applies with some minor variations to all cases.

Analysis of effects and verification of the proposed correction are in the left column, behavior of the equivalent wellblock radius on the right hand side. The first plot gives the wellbore pressure p_w along the well. The dimensionless well position $\frac{x}{L}$ is a running coordinate from the heel (0) of the well to the toe (1). The black solid and dashed lines are the results from the analytical and numerical solutions respectively. These lines appear in pairs for time of 1,2 and 4 days. Notice that the separation between each pair of lines closes with time, as the end of the transient flow regime is approached. This gap is what we are trying to correct for. In comparison to the infinite conductivity case (light grey, dashed lines) the impact of wellbore pressure drop becomes obvious. The drawdown is highest at the heel where the effect of friction is most pronounced. This effect is well captured by both the numerical and the semi-analytical solutions.

The second plot on the left gives the flow distribution along the well for a given time (3 days). As expected, rates at the tips are higher due to the increased exposure to the reservoir. The infinite conductivity solution is again symmetric.

Following the discussed procedure, the correct well index, here represented as $\ln r_{0n}$, is calculated. Plot (c) shows it as a function of time for three distinct position along the well. Two results become obvious. First, the correction (departure from conventional model) is highest in the transient flow regime and flattens out towards the $\ln r_{0n} = -2$ (Peaceman model) at later times. This corresponds to the observation in Plot (a) where the gap between numerical and analytical model closes with time. Second, heel and toe of the well require about the same level of correction while the middle of the well requires the highest amount of correction.

This symmetrical behavior is confirmed in Plot (d), where the normalized equivalent wellblock radius is given along the well length after 3 days. The dashed line gives again the conventional (constant) well model. The symmetry seen here is a little surprising considering the non-symmetric behavior due to wellbore pressure drop. But it makes sense when we recall that the friction correlations are applied to well pressures calculated by the well model and not before. A deviation from this can be seen in cases, where the analytical results are strongly affected by the accelerational part, which is not accounted for in the simulator. It is also interesting to note that the least correction is required at the tips of the well. It

would have been expected differently as the truly semispherical flow at the well ends is a clear violation of Peaceman's assumptions. But Eq. 2.12 together with Plot (b) show that the increased flowrate reduces the value for r_{0n} at the well ends.

Plots (c) and (d) can be summarized in a 3D plot. Fig. 2.6 shows $\ln r_{0n}$ vs. position on well and time. Corrections are highest at early times and in the middle of the well. The surface flattens out to the Peaceman solution with time.

The last step is to apply these results to a final simulator run. We prescribe new adjusted well transmissibilities onto our well for each wellblock and timestep. No segmentation is required, the first few timesteps will suffice (correction only in transient time). Returning to Plot (a) we can view the results in the form of a dotted line. The simulator reproduces the analytical solution. The dotted line overlies the solid one and they are hard to differentiate. Similar plots to Fig. 2.5 for the other five configurations in Fig. 2.2 are displayed in Figs. 2.7 to 2.11.

2.5.5 Impact of other parameters

Plots shown in Fig. 2.12 extract only the type (d) plot of Fig. 2.10. Recall the naming conventions: capitalized letters indicate a long well case (otherwise short well), number refers to isotropic permeability (in mD, no number: standard case of 3000 mD), "nf" means an infinite conductivity well (Table 2.2).

Plot (a) of Fig. 2.12 gives the correct r_{0n} for a long centered well after 3 days for different permeabilities. At high permeability, the well acts more like a point source than a line source. Reducing permeability by a factor of 10 to 300 mD, a constant plateau for the effective wellblock radius is formed, indicating a 2-dimensional, radial flow regime. At very low permeability, the conventional model becomes acceptable as a true radial flow regime is established. Plot (b) shows that a longer well deviates more substantially from the conventional well model than a short well does. This has probably to do with the fact that the reservoir is rather thin, so flow is far from being radial (rather linear) in the long well case.

The effect of well location is shown in (c) for long and in (d) for short wells. In both cases the closer the well gets to the boundary, the more the deviation from the standard well model.

Figure 2.5: Summary of Results for Case "EDGE"

Figure 2.6: Variation of $\ln r_{0n}$ as a Function of Time and Position along the Well

2.6 Conclusions

An analysis of well indices for a finite conductivity horizontal well is presented. Correct values were obtained from a semi-analytical solution in comparison with a numerical simulator. Well pressures allow for a relatively high tolerance on the correct value of the equivalent wellblock radius. The use of a numerical simulator with a conventional well model showed the following results:

- Considerations of frictional pressure drop in the well maybe important for horizontal wells and this option is already available in commercial simulators.
- Inflow (friction and acceleration) effects are neglected in commercial simulators, they become important in some cases, e.g., as the well gets shorter.
- Considerable errors are made by the numerical solution in the early time behaviour. This is mostly due to violations of the basic assumptions of the conventional well model.

A method for correcting the well index was presented. Several cases show an improvement in the numerical simulation, especially in the transient flow regime. Limitations of this procedure are due to dependence on an analytical solution for single phase flow in a box shaped reservoir.

Nomenclature

α = grid aspect ratio
 B, B_o = formation volume factor, $\frac{RB}{STB}$
 c_t = total compressibility, psi^{-1}
 c_ϕ = rock compressibility, psi^{-1}
 D = pipe diameter, ft
 ε = relative pipe roughness
 f = Fanning friction factor
 ϕ = porosity
 π = phase (subscript) or constant 3.14
 p^w = well perforation pressure, psi
 p_0 = well block pressure, psi
 k = permeability, mD
 σ absolute pipe roughness, ft
 k_r = relative permeability
 k_x, k_y, k_z = anisotropic k in directions x, y, z
 λ = mobility
 μ = viscosity, cp
 Re = Reynolds number
 r_0 = equivalent wellblock radius, ft
 r_{0n} = normalized r_0 , ft
 r_w = well radius, ft
 s = skin factor
 S, S_π, S_o = saturation
 T^w = well transmissibility
 WI = well index
 $\Delta x, \Delta y, \Delta z$ = block length in x, y, z direction, ft

References

- [1] Peaceman, D. W.: Interpretation of Well-Block Pressures in Numerical Reservoir Simulation, SPEJ, 183-94, June 1978.
- [2] Peaceman, D. W.: Interpretation of Well-Block Pressures in Numerical Reservoir Simulation with Nonsquare Grid Blocks and Anisotropic Permeability, SPEJ, 531-43, June 1983.

- [3] Babu, D. K. and Odeh, A. S.: Productivity of a Horizontal Well, SPERE, 417-241, November 1989.
- [4] Wolfsteiner, C. and Amado L.: Analytical Models for Different Flow Regimes in Horizontal Wells. Seminar Paper for the SPE Student Paper Contest, Vienna Basin Section, June 1995.
- [5] Penmatcha, V. R.: Modeling of Horizontal Wells with Pressure Drop in the Well. Dissertation, Dept. of Petroleum Engineering, Stanford University, 1997.
- [6] Colebrook, C. F.: Turbulent Flow in Pipes, With Particular Reference to the Transition Region Between the Smooth and Rough Pipe Laws, J. Inst. Civ. Eng., London, vol 11, 133-56, 1939.
- [7] Jain, A. K.: Accurate Explicit Equation for Friction Factor, Journal of Hydraulics Div., ASCE, vol 102, 674-677, 1976.
- [8] Haaland, S. E.: Simple and Explicit Formulas for the Friction Factor in Turbulent Pipe Flow, Trans. ASME, JFE, vol 105, p 89, 1983.
- [9] Chen, N. H.: An Explicit Equation for Friction Factor in Pipe, Ind. Eng. Chem. Fund., 18 (3), 296-97, 1979.
- [10] Eclipse Reference Manual, 96A Release, GeoQuest, Schlumberger.
- [11] Ouyang, L.-B., Arbabi, S. and Aziz, K.: General Wellbore Flow Model for Horizontal, Vertical and Slanted Well Completions, paper SPE 36608 presented at the SPE Annual Technical Conference and Exhibition, Denver, CO, October 6-9, 1996.

Figure 2.7: Summary of Results for Case "edge"

Figure 2.8: Summary of Results for Case "OFFCENTER"
48

OFFCENTER

Figure 2.9: Summary of Results for Case "offcenter"

49

offcenter

Figure 2.10: Summary of Results for Case "CENTER"

50

CENTER

Figure 2.11: Summary of Results for Case "center"

51

center

Figure 2.12: Impact of parameters on r_{on} (all plots at $t=3$ days)

3. Analysis of Gas Injection in Horizontal Wells

by Mahadevan Venkataraman, Sepehr Arbabi, and Khalid Aziz

Abstract

One of the major forces affecting the recovery of oil from reservoirs through horizontal wells is gravity. In this report, the effect of gas injection on the gravity drainage process in horizontal wells is discussed. The sweep efficiency of the gas injection process for different configurations of horizontal wells was studied using a CT scanner. The viscous and gravitational forces coming into play in the reservoir are discussed. The experimental setup for this project is given. The results from the experiments are analyzed and compared with the results from a numerical simulation of the same physical problem.

3.1 Introduction

Horizontal wells are used extensively in recovering oil from reservoirs as they expose a larger amount of the formation to the well than vertical wells. One of the important forces in the recovery of oil using horizontal wells is gravity. This is especially important in case of heavy oils where the reservoir pressure is not sufficient to move the viscous oil towards the wellbore. Gravitational forces are most effective when the horizontal well is placed close to the bottom of the reservoir and the distance for the flow of oil through the porous-medium is minimized.

Gas injection has been used widely to improve the recovery of oil. It can be used for maintaining pressure in the reservoir which in turn contributes to an increase in productivity or for miscible displacement. Nitrogen and carbon dioxide are the two commonly used non-hydrocarbon gases. In the following study, the effect of nitrogen injection on gravity drainage is studied and a sensitivity analysis of different factors affecting oil production is discussed.

3.2 Literature Survey

Gravity drainage in horizontal wells for the recovery of oil has been studied with regard to the optimal positioning of the well and its advantages over the use of vertical wells. Meszaros et al.[1] experimentally studied the drainage of oil into horizontal wells using inert

gas injection with oil viscosities varying from 1000 cP to 4000 cP and found that oil recovery increased from 50% to about 70% relative to the no gas injection case. The stability of the gas front was also studied for two horizontal wells placed one on top of the other. Gas was injected through the upper well and the stability of the front was studied using a physical model. It was also found that it is more difficult to maintain a stable gas front in a 3-D model than in a 2-D model. The production was mainly due to gravity drainage at low flow rates and the oil recovery was high. In all the experiments, the initial production rate was found to be high due to gravity drainage and as the front moved down, the effect of gravity reduced and hence the production of oil reduced considerably. At very low injection pressure, the production rate of oil was very slow but a steady gas front was maintained. Also as the injection pressure was increased, the recovery of oil decreased. Both carbon dioxide and nitrogen were used in the study and in both cases the dissolution of the gas in oil was neglected.

Dykstra and Dickinson[2] studied the effect of dip, reservoir thickness, oil mobility and well spacing on production and recovery from horizontal and vertical wells. In this study, mobility ratio was considered to be uniform for both the horizontal as well as the vertical well case even though this assumption is not valid. The head of the fluid flowing into the well was calculated as the height from the wellbore to the gas oil contact. This head and flow rate were found to decrease with time. In the case of horizontal formations (no dip), the ratio of horizontal and vertical well flow rates remained constant with time. It was found that for very thick formations, vertical wells performed better than horizontal wells. For inclined formations, horizontal wells were found to produce more than vertical wells. The thickness affected the flow rate ratios of horizontal to the vertical wells. Also, in case of vertical wells the well spacing was found to be important.

Williams[3] studied the effects of horizontal well placement and gravity on sweep efficiency for a gas injection process. In the cases considered here, the injection rate and the production rate were made equal. Two cases considered were staggered pattern horizontal wells and two parallel horizontal wells. A viscous to gravity ratio was defined and the pore volumes produced were evaluated as a function of this ratio. It was found that sweep efficiency of a staggered pattern was less than that of adjacent horizontal wells. Enhancement of the gravity drainage process through hydrocarbon additives were also studied by Nasr et al.[4]. The recovery of oil by the use of horizontal wells were found to be always greater than 50% and the addition of Naptha to the steam injection process increased the recovery to 82.5%. Extensive literature also exist on steam assisted gravity drainage and inert gas

injection[5-7].

In most of the above cases, the in-situ saturation at different instances of time were not identified. The experiments were done using oil with high viscosities. Also, the experiments were done with water wet rocks and the effect of gas injection was not studied in the oil wet rocks. The saturation profiles inside the models were not studied in these cases. The saturation profiles would indicate the regions where the oil recovery is high and where the residual oil saturations are high. In this report, the experimental setup to be used for studying the saturation profiles during the gravity drainage process using a CT scanner is discussed along with some simulation results based on the physical model.

3.3 Viscous and Gravitational Forces

In a gravity drainage process, the gravitational force is used to overcome the resistance to flow. The gravitational forces acting on the fluid are characterized by the gravity number which is the ratio of gravitational forces to viscous forces. On the other hand, capillary number is the ratio of capillary to viscous forces. The gravity and capillary numbers (Zhou et. al.[8]) are given as;

$$N_g = \frac{kA\Delta\rho g}{\mu_0 q_t}$$

$$N_c = \frac{kA\Delta p_c}{q_t \mu_0 L}$$

where k is the permeability of the medium, A is the cross-sectional area open to flow, $\Delta\rho$ is the difference in the densities of oil and gas, μ_0 is the oil viscosity, q_t is the total flow rate, and Δp_c is the capillary pressure between oil and gas at a given oil saturation.

In a gravity drainage process, the gravity number should be much greater than the capillary number. The flow rate in the model should be adjusted such that $N_g \gg N_c$. The gravity and capillary numbers are also used to scale up from the model to the field.

3.4 Experimental Setup and Procedure

The main aim of this study is to obtain sweep efficiencies for different horizontal well configurations. A Berea sandstone core (4.5 inches diameter and 1 foot long) was used for this study. The core was fired at 800 degrees Fahrenheit and was coated with epoxy. Firing the core reduces the obstruction due to clay swelling and passing oil through this fired core

Core Data	
Length	7.5"
Diameter	4.5"
Avg. Porosity	0.2
Avg. Permeability	300md

makes this an oil wet system. Inlet and outlet ports were provided on diagonally opposite ends of the core as shown in Figure 3.1. The ends of the core were sealed with a flow distributor. Decane was used for the initial case. The core was saturated with decane and the permeability of the core was measured. Injection flow rates were measured using a gas flow meter, and for doing constant flow rate experiments, a mass flow controller was used to regulate the injection rate. The oil coming out of the collection port was measured at different times in order to obtain the oil production and a gas flow meter was also provided at the outlet in order to measure gas production after breakthrough. Also, two 1 inch diameter cores of 1.5 inches length were obtained from a similar Berea sandstone. The cores were obtained from perpendicular axis in order to check for the anisotropy. Permeability was calculated using the gas permeameter.

The gravity drainage process by itself would take a very long time. This is because as the height of the fluid decreases, the pressure head drops and it takes a very long time to drain the oil out. Instead, gas was injected at a very low flow rate to speed up the recovery. Care was taken to make sure that flow was dominated by gravitational forces. The gas used in this case was Nitrogen. The injection rate used for nitrogen is 120 cc/hr. At this flow rate, the gravity number was found to be larger than the viscosity number.

A CT scanner was used to scan the core for saturations and the saturation profiles was obtained at different times and at different depths of the cores. By monitoring the oil saturation values, the path of the injected gas can be tracked through the system.

3.5 Simulation of the experiment

Experiments were simulated with the help of a commercial simulator (Eclipse 96a). A lab scale study was done in order to obtain the sensitivity of different parameters to production and also to match the production history from the experiments. Also, the effects of simulating a horizontal well using a number of inlet and outlet ports were studied using this model. The size of the core was 4.5 inches in diameter and 30 cms in length. This

Figure 3.1: Experimental Set-up

was modeled using cubic grids of 0.5 cm x 0.5 cm x 0.5 cm each and no flow boundary conditions were applied to the boundary blocks. Transmissibilities of some boundary blocks were set to zero in order to approximate the cylindrical shape of the core. Cross section of the simulation model is shown in Figure 3.2. A horizontal injector was completed at the top and a horizontal producer was used at the bottom. The producing well was kept at atmospheric pressure and conditions for the injection wells were either constant flow rate or constant pressure. The permeability of the core used was 400 mD and the porosity values for the core varied between 0.1 and 0.3. Porosity value for the simulations were taken to be 0.2. The pore volume of the simulation model approximately was equal to the pore volume of the core. Typical Corey type relative permeabilities for Berea sandstone were used. The experiments were simulated using different permeabilities and injection conditions. A typical set of relative permeability vs saturation curves used for this study is given in Figure 3.3. The oil production rate for horizontal injector placed at the top and horizontal producer placed at the bottom is given in Figure 3.4. The shape of the production curve obtained was found to match with that of the general trend of production curves in gravity drainage processes. There is a sharp increase in the production rate at early times and as expected, production falls off with time.

3.6 Sensitivity Analysis

Some of the important parameters that affect oil recovery in horizontal wells are injection conditions, permeability, anisotropy, relative permeabilities and well configurations. Figure 3.5 shows the effect of gas injection rate. As the flow rate was increased, the oil production curves also shifted upwards and gas breakthrough time decreased. But no significant increase in the total oil recovery was observed. Permeability was varied keeping all the other conditions constant and the effect of permeability on oil production is shown in Figure 3.6. As the permeability is decreased, the oil production curve shifts downwards indicating lower recovery at a given time. The effect of anisotropy was also studied by varying the vertical permeability of the system. Figure 3.7 shows that anisotropy affects the gas breakthrough time the most. But no significant changes were observed in the total recovery. The effect of relative permeability is more pronounced after gas breakthrough. Lowering the end point relative permeability of oil decreased the total oil recovered. Figure 3.8 shows the effect of well configuration on oil production. Three cases were considered: vertical injector and vertical producer, parallel horizontal injector and horizontal producer and staggered horizontal injector and producer. The recovery was found to be greater in the horizontal well cases

Figure 3.2: Cross-section of the simulation model near injector well showing pressure gradient just before the strat of simulation

Figure 3.3: Corey-type relative permeability curve with $n = 2$; $k_{ro_{max}} = 0.3$ and $k_{rg_{max}} = 0.7$. This curve was used to match the experimental results for the vertical case

Figure 3.4: Simulation results for a vertical well case with $k = 300md$; Corey-type relative permeabilities; injection rate of 120 cc/hr

Figure 3.5: Effect of injection rate on cumulative production with all the other parameters fixed (simulation)

and the highest recovery was observed in the staggered horizontal injector and producer. Figures 3.9-3.11 show the typical saturation profiles at the end of the experiment for these three cases. This sensitivity analysis helps in identifying the key parameters for matching the experimental data with the simulation.

3.7 Experimental Results

Two sets of experiments were conducted. In one set, a constant gas injection rate was used to study oil production for three different well configurations. Nitrogen was injected at 120 cc/hr in all three cases. Figure 3.12 shows the comparison of the oil production in three cases. oil production follows the general trend shown in the sensitivity analysis. Figures 3.13-3.15 show typical saturation profiles obtained from the CT scanner at different times for the three cases. In the second set, one of the experiments was repeated to check if the experiments were repeatable. The vertical well case with a constant injection rate of 120 cc/hr was repeated and the results were found to match the previous experiment within reasonable limits (Figure 3.16). Two experiments were also performed under constant injection pressure condition of one psig. The well configurations used for this were horizontal wells (staggered and parallel). The saturation profiles were found to be similar to constant injection conditions. A mass balance calculation was performed in all the above cases and the errors were mainly due to interpolation of the saturations between two slices. The trend in saturations on the whole was found to match with the saturations obtained from simulations.

Some of the difficulties faced while doing the experiments were fracturing of the core while firing and core resaturation. The core was gradually heated to 800 degrees Fahrenheit and was cooled very slowly in order to avoid cracking of the core. When the core was resaturated, air was trapped in the core due to by-passing of decane in certain regions. In order to avoid this, the core was vacuumed for a long time and decane was allowed to flow in from the bottom of the core. This reduced the problem of gas getting trapped in the core.

3.8 Matching Experimental Data with Simulation

The results for experiments were compared with results from simulations. Cumulative oil production obtained from the experiment was compared with the the oil production from the simulator for similar conditions. Parameters like end-point relative permeabilities and vertical permeabilities were changed in order to match the simulator and experimental results. Other parameters which were also considered for obtaining the history match were

Figure 3.6: Effect of horizontal permeability on cumulative production with all the other parameters fixed (simulation)

Figure 3.7: Effect of vertical permeability on cumulative oil production with all the other parameters fixed

Figure 3.8: Simulation results comparing cumulative productions for the three well configurations with $k = 300$ md and gas injection rate of 120 cc/hr

Figure 3.9: Saturation contours from the simulator model after 8 hrs of injection through vertical injector and producer at the injection rate of 120 cc/hr

Figure 3.10: Saturation contours for staggered horizontal injector and producer after 8 hrs of gas injection at the rate of 120 cc/hr

Figure 3.11: Saturation contours for parallel horizontal injector and producer after 8 hrs of gas injection at the rate of 120 cc/hr

the initial saturation and capillary pressure curves. Results from the sensitivity analysis were used in order to change the parameters in the simulation. The cumulative oil produced versus time is compared for the simulator and the experimental results (Figure 3.17). A reasonable match between the data is obtained and the end point relative permeabilities that are used here are 0.3 for oil and 0.6 for gas and Corey type relative permeabilities (with power $n = 2$) are used for the rest of the saturation. The vertical permeability used for the simulation was 150 md which was found to be close to the vertical permeability of the core. After a match in the production history was found, the pressure in the injection port was compared with the bottom hole pressure of injection well in the simulation (Figure 3.18) and the saturation profiles were compared with the CT images (Figure 3.19). Similar procedure is being applied to the horizontal well cases for obtaining the history match.

3.9 Conclusions

It was observed that gas injection at low flow rates aids the gravity drainage process and increases oil production rate. In all of the experiments, the gas was found to stay at the top of the model and the oil was forced towards the outlet which is at the bottom of the reservoir. The amount of oil-recovered by this process is dependent upon the well configuration and gas injection rate. In case of parallel horizontal wells, the gas breakthrough occurs at an early stage and the recovery is reduced. The experimental results were found to match with the simulation results especially in case of cumulative oil production and injection pressure. An exact match in the saturation profile might not be possible due to heterogenities in the Berea sandstone. Therefore, only the average saturation over a region was matched with the simulation.

Figure 3.12: Comparison of experimental cumulative oil production for the three well configurations at the constant injection rate of 120 cc/hr

Figure 3.13: Cross-sectional saturation profiles for the vertical well case at various values of PVI with constant injection rate of 120 cc/hr . The left column is about 2 inches, the middle column is about 4 inches, and the right column is about 6 inches away from the injection port

Figure 3.14: Same as 3.13 but for the parallel horizontal well configuration

Figure 3.15: Same as 3.13 but for the staggered horizontal well configuration

Figure 3.16: Test for repeatability of experimental results for the vertical well configuration

Figure 3.17: Matching cumulative oil production history from the experiment and simulation for the vertical well case. Parameters used in the simulation are: $k_z = 150md$ and no capillary pressure. Injection rate in both cases is fixed at 120 cc/hr

Figure 3.18: Matching injection pressure history from the experiment and simulation for the vertical well case. Same parameters as in 3.17 were used.

Figure 3.19: Gas saturation profiles from simulations at 2 inches, 4 inches, and 6 inches from the end of the core. The cross sections are for 0 PVI (left column), 0.4 PVI (middle column), and 2.0 PVI (right column) for the vertical well configuration

Reference

1. Meszaros, G., Chakma, A., and Islam, M.R., "Scaled model studies and numerical simulation of inert gas injection with horizontal wells", SPE 20529, 65th annual technical conference and exhibition, New Orleans, LA, Sept 23-26, 1990, pg 585.
2. Dykstra, H., and Dickinson, W., "Oil recovery by gravity drainage into horizontal wells compared with recovery from vertical wells", SPE 19827, 64th annual technical conference, San Antonio, TX, Oct 8-11, 1989, pg 607.
3. Williams, Richard Jr., "The effects of horizontal well placement and gravity on sweep efficiency", Ch 4, M.S. Thesis, 1997, pg 23.
4. Nasr, T.N., Kimber, K.D., Vondrinsky, D.A., and Jha, K.N., "Process enhancement in horizontal wells through the use of vertical drainage channels and hydrocarbon additives", SPE 21794, Western regional meeting, Long Beach, CA, Mar 20-22, 1991, pg 417.
5. Jha, K.N., and Chakma, A., "Nitrogen injection with horizontal wells for enhancing heavy oil recovery: 2-D and 3-D models", SPE 23029, Asia-Pacific conference, Perth, Australia, Nov 4-7, 1991, pg 765.
6. Edmunds, N.R., and Sugget, J.C., "Design of a commercial SAGD heavy oil project", SPE 30277, Heavy Oil symposium, Calgary, Canada, 19-21 June 1995, pg 287.
7. Ahner, P.F., and Sufi, A.H., "Physical model steamflood studies using horizontal wells", SPE 20247, SPE Reservoir Engineering, Feb. 1994, pg 59.
8. Zhou, D., Fayers, F.J., and Orr, F.M. Jr., "Scaling of Multiphase flow in simple heterogeneous porous media", SPE 27833, pg 559.

4. Multiple Solutions for Separated Gas-Liquid Flow in a Wellbore

by Liang-Biao Ouyang and Khalid Aziz

Abstract

Multiphase pipe flow is an important consideration for horizontal wells. Multiple solutions are found for separated gas-liquid flow in pipes. Occurrence of multiple solution is shown and briefly discussed. The region of multiple solution is displayed in both the U_{SL} - U_{SG} map and the X - Y map. Physical considerations and numerical simulations of the transient process for separated gas-liquid flow in a pipe has lead to the selection of correct solution.

4.1 Introduction

Mathematically multiple solutions can happen for separated upward gas-liquid two-phase flow, such as stratified smooth flow, stratified wavy flow, annular flow, and annular-mist flow, as found by several researchers, including Baker & Gravestock [1], Landman [2, 3], Barnea & Taitel [4], Petalas & Aziz [5]. Usually three solutions for liquid holdup (liquid height) and thus pressure gradient can be obtained under certain flow conditions. To make things worse, it is not uncommon to have a significant difference between the highest and the lowest solutions for the liquid holdup. Although most researchers agree that nonunique solutions do exist for separated upward flow, there is still discrepancy in regard to the solution(s) which is/are possible in practice. For example, Landman [2, 3] stated that both the lowest and the intermediate solutions are physically possible, while Barnea & Taitel [4] argued that only the lowest solution is reasonable.

Wellbore flow differs from regular pipe flow in that there exists either wall inflow for production wells or wall outflow for injection wells. In this chapter, similar approaches as used by above-mentioned researchers will be applied for gas-liquid separated wellbore flow to study the occurrence of nonunique solutions and to identify the physically realistic solution(s). Due to the similarity between stratified flow and annular flow, emphasis here will be on multiple solution issues related to stratified flow. However, analyses and observations for stratified flow are expected to be also valid for annular and annular-mist flows.

4.2 Occurrence of Multiple Solutions

For steady-state gas-liquid stratified flow in pipes where no wall mass transfer occurs, Baker & Gravestock [1] were the first to find nonunique solutions for liquid height and liquid holdup for upward flow. The phenomena is clearly seen in Fig. 4.1 and Fig. 4.2, which display liquid height and pressure gradient as a function of the Lockhart-Martinelli parameter X for air-water stratified flow in a 2.047inch ID pipe. Fig. 4.1 and Fig. 4.2 show that dimensionless liquid height h_L/D and absolute pressure gradient dp/dx increase with the Lockhart-Martinelli parameter X for $Y = -100, 0, 100$, and 10000. For a specific value of X , there is only one liquid height and one pressure gradient associated with it. Besides, pressure gradient curves become parallel to each other for different Y values when the Lockhart-Martinelli parameter X becomes large (Fig. 4.2). An explanation for this is given below as a footnote*.

Recall that the Lockhart-Martinelli parameter X is defined as the ratio between superficial pressure gradient of liquid phase and that of gas phase. If pipe geometry, pipe inclination angle, fluid properties, and flow rate for one of the two phases are fixed, then specifying a X value is equivalent to fixing the flow rate for the second phase, and thus the whole system becomes fully defined. Under these conditions, liquid holdup and pressure gradient should also be fixed. Physically this makes sense, because there is only *one* liquid height and *one* pressure gradient expected for each set of flow rates if all the other parameters for the system are specified. However, mathematically this is not always the case. Look again at both Fig. 4.1 and Fig. 4.2, it is observed that *three* rather than one solution for liquid

*Along each pressure gradient curve, pipe inclination angle, inclination parameter Y and superficial gas velocity U_{SG} are specified. The only variable parameter is superficial liquid velocity U_{SL} . When the X value gets high, flow in the pipe becomes close to single phase liquid flow. Moreover, liquid flow rate will be high due to the fact that the value of X is large. For single phase liquid flow

$$-\frac{dp}{dx} = \rho_L g \sin \theta + \frac{2f_L \rho_L U_L^2}{D} = -\left(\frac{dp}{dx}\right)_g - \left(\frac{dp}{dx}\right)_f \quad (4.1)$$

At high liquid flow rates, $-\left(\frac{dp}{dx}\right)_g \ll -\left(\frac{dp}{dx}\right)_f$, $h_L/D \rightarrow 1$, $U_{SL} \rightarrow U_L$, and $f_{SL} \rightarrow f_L$, so

$$-\frac{dp}{dx} \approx \frac{2f_L \rho_L U_L^2}{D} \approx \frac{2}{D} \rho_L f_{SL} U_{SL}^2 = \frac{2}{D} f_{SG} U_{SG}^2 \rho_G X^2 \quad (4.2)$$

hence

$$\log \left| \frac{dp}{dx} \right| \approx 2 \log X + \log \left[\frac{2}{D} f_{SG} \rho_G U_{SG}^2 \right] = 2 \log X + \log [f(U_{SG})] \quad (4.3)$$

Eq. 4.3 indicates that the slope of the pressure gradient vs X curve on log-log plot will be equal to 2, while the curve location (or the intercept) depends on the superficial gas velocity or gas flow rates.

height and pressure gradient exist in certain X range for cases of $Y = -5$ and $Y = -10$. Since only one solution is physically possible, it is imperative to determine which solution of liquid height and pressure gradient is the *real* and *correct* one.

Figure 4.1: Liquid Height Variation with Lockhart-Martinelli Parameter X in an Air-Water System ($f_i = f_{wG}$)

4.3 Parameters Affecting Multiple Solutions

Empirical correlations are required for wall friction factors for both gas and liquid phases and for interfacial friction factor in order to complete a stratified flow model. Different correlations have been reported in the literature for evaluating these factors. For example, more than 20 correlations for interfacial friction factor have been proposed (Ouyang, [6]). It is thus natural to first study possible effects of correlations applied in the model on multiple solutions. Figs. 4.3 and 4.4 show comparison of liquid height and pressure gradient for $Y = -5$ by using two different interfacial friction factor correlations, the Duns & Ros correlation [7] modified by Baker et al. [8] and a simple correlation $f_i = f_{wG}$ as first proposed by Taitel & Dukler [9].

Both liquid height and pressure gradient plots indicate that multiple solutions only occur for the $f_i = f_{wG}$ correlation but vanish for the modified Duns & Ros correlation. This can be explained by the X - Y map for multiple solution to be discussed later in this

Figure 4.2: Pressure Gradient Variation with Lockhart-Martinelli Parameter X in an Air-Water System ($f_i = f_{wG}$)

Figure 4.3: Liquid Height Variation with Lockhart-Martinelli Parameter X in an Air-Water System ($Y = -5$)

Figure 4.4: Pressure Gradient Variation with Lockhart-Martinelli Parameter X in an Air-Water System ($Y = -5$)

chapter. Note that little difference is found between the predicted liquid height and pressure gradient for the two interfacial friction correlations in low and high X regions where gas-liquid stratified flow evolves into single phase gas flow and single phase liquid flow, respectively.

One important issue related to the nonuniqueness of liquid holdup and pressure gradient is the range of parameters in which multiple solutions exist. Different parameters, including pipe inclination angle, superficial gas velocity, superficial liquid velocity, pipe diameter, and so on, have been investigated in depth. Two or more correlations for the interfacial friction factor were applied in all cases. Part of the results thus obtained are displayed in Figs. 4.5, 4.6, and 4.7.

Range of pipe inclination angle in the multiple solution region is significantly affected by the interfacial friction factor correlation used in a stratified flow model (Fig. 4.5). For the modified Duns & Ros correlation, the range of inclination angle in the multiple solution region is about $8.26^\circ \sim 16.04^\circ$, whereas the range becomes around $12.12^\circ \sim 58.0^\circ$, $5.41^\circ \sim 13.81^\circ$ and $14.42^\circ \sim 90.0^\circ$ respectively, for $f_i = 0.0142$, $f_i = f_{wG}$, and the Cohen & Hanratty [10] correlation[†]. There exists no pipe inclination angle at which multiple solution appears if the Andritsos & Hanratty [11] correlation is applied, because superficial gas velocity in

[†]Stratified flow may not exist for inclination angle greater than a certain value, which is believed to be less than 30° .

this case is more than ten times higher than the transition value $U_{SG,T}$. As a consequence liquid holdup must be very small (less than 0.01) in order to avoid extremely high value of interfacial friction factor and inconsistency in gas and liquid phase momentum balance equations.

By varying superficial gas velocity, multiple solutions are found for all five interfacial friction factor correlations (Fig. 4.6). Ranges of superficial gas velocities in which multiple solutions are obtained are given in Table 4.1. The range for different correlations are substantially different.

Table 4.1: Range of Superficial Gas Velocity for the Nonuniqueness of Liquid Height and Pressure Gradient Solutions

Correlation	Range of Superficial Gas Velocity (<i>ft/sec</i>)
Modified Duns & Ros (1963)	82.56 – 113.20
$f_i = 0.0142$	54.96 – 90.92
$f_i = f_{wG}$	88.16 – 136.96
Cohen & Hanratty (1965)	35.32 – 83.44
Andritsos & Hanratty (1987)	39.48 – 43.76

Range of ID's over which multiple solutions are obtained are relatively small (Fig. 4.7). The nonuniqueness of solutions happens at a pipe internal diameter of around 2.0 *inch*.

It needs to be mentioned that the nonuniqueness of solutions may result in computational problems for practical design of wells and pipelines. Fig. 4.8 shows pressure gradient as a function of pipe diameter with $Q_L = 0.005 \text{ ft}^3/\text{sec}$ and $Q_G = 2.765 \text{ ft}^3/\text{sec}$. Multiple solutions appear in about a range of pipe ID from 2.0 *inch* to 2.5 *inch*. Within this range (although it is a small one), there are *two* pipe internal diameter for which identical pressure gradients are obtained.

4.4 Multiple Solution Region: X-Y Map

In Figs. 4.5, 4.6, and 4.7, only the range(s) of multiple solution for *one* parameter of interest can be demonstrated. It is more useful if ranges for many parameters can be shown on a single plot. Fig. 4.9, an improvement over Figs. 4.5, 4.6, and 4.7, identifies the ranges for both superficial gas velocity and superficial liquid velocity where multiple solutions exist. The observation that interfacial friction factor correlation significantly affects the region of multiple solutions as indicated by Figs. 4.5 and 4.6 is again clearly demonstrated here. In

Figure 4.5: Liquid Height Expressed as a Function of Pipe Inclination Angle for an Air-Water System ($U_{SL} = 0.01 \text{ ft/sec}$, and $U_{SG} = 100 \text{ ft/sec}$)

Figure 4.6: Liquid Height Expressed as a Function of Superficial Gas Velocity for an Air-Water System ($U_{SL} = 0.01 \text{ ft/sec}$, and $\theta = 10^\circ$)

Figure 4.7: Liquid Height Expressed as a Function of Pipe Internal Diameter for an Air-Water System ($Q_L = 0.005 \text{ ft}^3/\text{sec}$, $Q_G = 2.765 \text{ ft}^3/\text{sec}$, and $\theta = 10^\circ$)

Figure 4.8: Pressure Gradient Variation with Pipe Internal Diameter ($Q_L = 0.005 \text{ ft}^3/\text{sec}$, $Q_G = 2.765 \text{ ft}^3/\text{sec}$, and $\theta = 10^\circ$)

addition, Fig. 4.9 shows that superficial liquid velocity is very low in the multiple solution region (less than 0.1 *ft/sec*) and it is about 2 to 3 orders smaller than the superficial gas velocity. This conclusion is very enlightening for the stability analysis discussed later in the Section 4.7.

Figure 4.9: Region for the Solution Nonuniqueness (No Wall Influx, $\theta = 10^\circ$)

Although Fig. 4.9 does provide more information about the multiple solution region, the region is apparently dependent on pipe geometry, fluid types, fluid properties and other parameters. In other words, the multiple solution region will alter once any one of the parameters, such as pipe internal diameter, gas density, gas viscosity, liquid density, and so on, is changed. So rather than using dimensional variables as the abscissa and the ordinate, dimensionless variables are preferred and make more sense. Fig. 4.10 is an effort towards this objective (Landman [2, 3]). This figure is the most general map (called *X-Y* map) for determining the parameter ranges of multiple solutions. Since it is represented by two dimensionless variables, *X* and *Y*, it is anticipated that the map should be applicable to different fluid system and different pipe geometry, provided that the empirical closure relationships needed for the stratified flow model are identical. The observation is verified by Figs. 4.11 and 4.12. The pipe inclination angle and the pipe internal diameter were chosen as 10° and 2.047inch in creating Fig. 4.10. A pipe inclination angle of 1° and a pipe internal diameter of 6.18inch are also considered in Fig. 4.11 and Fig. 4.12. Only a slight variation in the

multiple solution region is observed in the X - Y maps, which implies that *the X - Y map can indeed be used to approximately determine solution nonuniqueness for different fluid system and different pipe geometries.*

Fig. 4.10 is quite illuminating and has different applications. For example, it can be used to explain the results shown in Figs. 4.1, 4.3, 4.5 and 4.9. If the modified Duns & Ros [8] correlation is used, the maximum inclination parameter Y in the multiple solution region is about -5.42 (Fig. 4.10), whereas it increases to -3.71 for the $f_i = f_{wG}$ correlation. This observation is consistent with Fig. 4.3. In Fig. 4.3, $Y = -5$ is even larger than the maximum Y for solution nonuniqueness for the modified Duns & Ros correlation [7], hence multiple solutions are not observed. In Fig. 4.3, $Y = -5$ is lower than the maximum Y for the $f_i = f_{wG}$ correlation, hence nonunique solutions appear. Other possible applications of the X - Y map will be discussed later.

Figure 4.10: Region of Solution Nonuniqueness in a X - Y Map (No Wall Inflow/Outflow)

4.5 Effects of Wall Inflow/Outflow

As indicated earlier in this chapter, gas-liquid two-phase pipe flow with mass transfer through the pipe wall is the primary issues of interest. In the following paragraphs, effects of wall mass transfer on multiple solution behavior will be explored.

Figure 4.11: Dependence of the Region of Solution Nonuniqueness on Pipe Inclination Angle in a X - Y Map (No Wall Inflow/Outflow, Modified Duns & Ros Correlation Used)

Figure 4.12: Dependence of the Region of Solution Nonuniqueness on Pipe ID in a X - Y Map (No Wall Inflow/Outflow, Modified Duns & Ros Correlation Used)

We start with the case of radial gas inflow. As shown by Figs. 4.13 and 4.14, if axial gas and liquid flow rates, fluid properties and pipe geometry are identical, as those in Figs. 4.1 and 4.2, while inflow parameter I as defined in Ouyang [12] is set to be 10, (where positive I corresponds to gas inflow, or more accurately, gas inflow dominant case), multiple solutions for both liquid height and pressure gradient disappear for all the six inclination parameter Y s displayed. But this only means that multiple solutions vanish for the Y values investigated and it does not imply that solutions will be unique for all other Y values. On the contrary, nonuniqueness in liquid height and pressure gradient still exists for other Y values as will be shown later in Fig. 4.17.

For the liquid inflow case shown in Figs. 4.15 and 4.16, with inflow parameter I equal to -10 , multiple solutions for liquid height and pressure gradient occur not only for inclination parameter $Y = -5$ and $Y = -10$, but also for $Y = 0$ (horizontal pipe). This shows that nonuniqueness may also exist for horizontal and even downward flow if the liquid influx is strong enough.

Figure 4.13: Liquid Height Variation with the Lockhart-Martinelli Parameter X for Gas Dominant Inflow Case ($f_i = f_{wG}$, $I = 10$)

Figs. 4.13, 4.14, 4.15 and 4.16 should be analyzed with caution, otherwise, incorrect and misleading conclusions about multiple solution can be drawn. A better approach is to use the X - Y map similar to that shown in Fig. 4.10. Fig. 4.17 is a X - Y map showing multiple solutions for liquid height and pressure gradient for gas inflow dominant case with

Figure 4.14: Pressure Gradient Variation with the Lockhart-Martinelli Parameter X for Gas Dominant Inflow Case ($f_i = f_{wG}$, $I = 10$)

Figure 4.15: Liquid Height Variation with the Lockhart-Martinelli Parameter X for Liquid Dominant Inflow Case ($f_i = f_{wG}$, $I = -10$)

Figure 4.16: Pressure Gradient Variation with the Lockhart-Martinelli Parameter X for Liquid Dominant Inflow Case ($f_i = f_{wG}$, $I = -10$)

$I = 10$. Fig. 4.18 shows the same for a liquid inflow dominant case with $I = -10$. Two different interfacial friction factor correlations are applied in obtaining these X - Y maps. By comparing these figures with the X - Y map shown in Fig. 4.10, it is easily seen that the multiple solution regions alter due to both gas and liquid influx. However, the overall appearance of the region does not change significantly, and the whole region seems to move in the direction of decreasing Y for gas inflow dominant case, but move in the opposite direction for liquid inflow dominant case. Amount of the shift in the region is approximately equal to the absolute value of the inflow parameter I . For example, if the modified Duns & Ros correlation is used for the interfacial friction factor evaluation, the right hand side boundary of the multiple solution region is approximately represented by $Y = -5.42$ in Fig. 4.10, the same boundary becomes $Y = -16.07$ for the liquid inflow dominant case (Fig. 4.18) and $Y = +4.61$ for the gas inflow dominant case (Fig. 4.17).

Because of the shift in multiple solution region, it is no longer true that solution nonuniqueness only exists for upward flow situations where Y is negative. Multiple solutions are also expected for horizontal and even downward flow cases, as those shown in Figs. 4.15 and 4.16 where liquid inflow is dominant. On the other hand, considering the fact that normally stratified flow only exists for horizontal and downward flow, and for upward flow with a small inclination angle, multiple solution may disappear for gas inflow dominant case

provided that the inflow parameter I is so high that the maximum inclination parameter Y for solution nonuniqueness becomes less than the minimum Y a system may achieve in order to maintain stratified flow.

Figure 4.17: Multiple Root Region for Gas Dominant Inflow Case ($I = 10$)

4.6 Multiple Solution Analysis via Interfacial Friction Shear

Another approach to study multiple solutions for liquid height and pressure gradient is by using the concept of interfacial shear stress. There are two different ways to evaluate the interfacial friction shear stress, one is to compute it by the momentum balance equations, and the other is to calculate it by applying the definitions associated with certain interfacial friction factor correlation. The two shear stresses thus evaluated are termed as τ_{imb} and τ_{idf} , respectively. Interfacial friction shear stress based on the momentum balance equations, τ_{imb} , does not depend on an interfacial friction correlation, while the τ_{idf} is primarily a function of the correlation, together with other parameters, including gas and liquid flow rates as well as liquid holdup or liquid height. If all the other parameters except liquid height (or liquid holdup) are fixed, the liquid holdup or liquid height which satisfies the condition that two interfacial friction shear stresses are identical, or, $\tau_{idf} = \tau_{imb}$, will be the liquid holdup for the system for this interfacial friction factor correlation. If different correlations are used,

Figure 4.18: Multiple Root Region for Liquid Dominant Inflow Case ($I = -10$)

the liquid holdup obtained may not be the same. In reality, only one liquid holdup can be correct. So the following criterion can be used to evaluate the appropriateness of an interfacial friction factor correlation, *the closer the predicted liquid holdup or liquid height to the measured or real value, the better the correlation*. Detailed evaluation and comparison of different interfacial friction factor correlations has been performed by Ouyang [6] and it was found that no single correlation can be satisfactorily applied for all situations.

Although the predicted liquid holdup or liquid height will be correlation dependent, it is anticipated that the primary features should not vary substantially if different correlations are applied. This may be seen in Figs. 4.19 and 4.20 where the Duns & Ros interfacial friction factor correlation and $f_i = f_{wG}$ were used. In Fig. 4.19, there exist three interception points between the τ_{idf} vs h_L/D curve and the τ_{imb} vs h_L/D curve for $U_{SL} = 0.001 \text{ ft/sec}$ and $U_{SL} = 0.01 \text{ ft/sec}$. Therefore, multiple solutions occur for these two superficial liquid velocities. On the other hand, there is only one interception point between two curves for $U_{SL} = 0.1 \text{ ft/sec}$ and $U_{SL} = 1.0 \text{ ft/sec}$, which means that solutions for liquid height and pressure gradient are unique.

Fig. 4.19 also shows that the interfacial friction shear stress based on momentum balance relationships τ_{imb} must have a minimum for situations with multiple liquid height solutions. Hence, *One necessary condition for solution nonuniqueness in liquid height, liquid*

holdup and pressure gradient is that the interfacial friction shear stress based on the momentum balance equations has a local minimum. Similar conclusion can be drawn based on Fig. 4.20 where $f_i = f_{wG}$ was used.

Figure 4.19: Interfacial Friction Shear Stress Evaluated Based on Momentum Balance and from Definition (No Wall Inflow/Outflow, $U_{SG} = 100 \text{ ft/sec}$)

Influence of liquid or gas inflow on the gas-liquid stratified flow can also be demonstrated by the τ_i-h_L/D plots (Figs. 4.21 and 4.22). For liquid inflow as shown in Fig. 4.21, multiple solution only happens for $\bar{U}_{SL} = 0.001 \text{ ft/sec}$ but vanishes for $U_{SL} = 0.01 \text{ ft/sec}$. This is expected from the X - Y map shown in Fig. 4.18. Although the (X, Y) coordinate of the system stays the same, it falls outside the multiple solution region for liquid inflow dominant case ($I = -10$) due to the shift of the region towards larger Y values. Local minimum in interfacial friction shear stress based on momentum balance still exists for $U_{SL} = 0.01 \text{ ft/sec}$, hence the existence of a local minimum is only a *necessary* rather than *sufficient* condition for solution nonuniqueness.

For gas inflow dominant case with $I = 10$, the local minimum in the interfacial friction shear stress disappears (Fig. 4.22), hence, no multiple solution exists. The observation can also be explained by using the X - Y map shown in Fig. 4.17.

Figure 4.20: Interfacial Friction Shear Stress Evaluated Based on Momentum Balance and from Definition (No Wall Inflow/Outflow, $U_{SG} = 100$ ft/sec)

Figure 4.21: Interfacial Friction Shear Stress Evaluated Based on Momentum Balance and from Definition ($I = -10$, $U_{SG} = 100$ ft/sec)

Figure 4.22: Interfacial Friction Shear Stress Evaluated Based on Momentum Balance and from Definition ($I = 10$, $U_{SG} = 100 \text{ ft/sec}$)

4.7 Physical Validation of Multiple Solutions

As shown in previous sections, there may exist up to three solutions of liquid height (liquid holdup) which satisfy steady-state gas and liquid momentum balance equations. Naturally a question is raised: Which solution(s) is/are physically realistic? One way to answer this question is to conduct a stability analysis for average liquid holdup (liquid height).

Based on the X - Y map (Figs. 4.10, 4.18 and 4.17), it is seen that liquid velocity is much lower than gas velocity in the region of multiple solutions, because the Lockhart-Martinelli parameter X is very small in the region. Therefore, it is reasonable to assume that the dynamic response to variables for gas phase is very rapid and gas flow can be treated as quasi-steady state with respect to any time interval, and the momentum balance equation for the gas phase (Ouyang [12]) changes to

$$\frac{dp}{dx} + 2\rho_G U_G \frac{q_{IG}}{A_G} = -\frac{\tau_i S_i}{A_G} - \frac{\tau_{wG} S_G}{A_G} - \rho_G g \sin \theta \quad (4.4)$$

In addition, assume that liquid height is nearly uniform along a small segment of the pipe under consideration, as a consequence, average liquid holdup and average liquid velocity will primarily depend upon time t rather than axial location x . Under these assumptions,

transient mass and momentum balance for the liquid phase (Ouyang [12]) can be simplified to

$$\frac{dE_L}{dt} = \frac{1}{l} (U_{SL} - E_L U_L) + \frac{q_{IL}}{A} = f_1(E_L, U_L) \quad (4.5)$$

$$\frac{dp}{dx} + \rho_L \frac{\partial U_L}{\partial t} + 2\rho_L U_L \frac{q_{IL}}{A_L} = \frac{\tau_i S_i}{A_L} - \frac{\tau_{wL} S_L}{A_L} - \rho_L g \sin \theta \quad (4.6)$$

Eliminating the dp/dx term from Eqs. 4.4 and 4.6 yields

$$\frac{dU_L}{dt} = \frac{1}{\rho_L} K(E_L) (\tau_{idf} - \tau_{imb}) = f_2(E_L, U_L) \quad (4.7)$$

where

$$K(E_L) = \frac{S_i}{A_L} + \frac{S_i}{A_G}$$

In order to obtain physically stable solutions (with respect to time) for both average liquid velocity and average liquid holdup (i.e., liquid height), two eigenvalues of matrix $\partial(f_1, f_2)/\partial(E_L, U_L)$ at steady state must be both negative (Barnea & Taitel [13]).

Applying the steady-state relationships $\tau_{idf} = \tau_{imb}$ and $U_{SL} = U_L E_L$ and noting the τ_{idf} definition, the following eigenvalue equation can be obtained

$$\lambda^2 - \left(-\frac{U_L}{l} + K \frac{\partial(\tau_{idf} - \tau_{imb})}{\partial U_L} \right) \lambda + \frac{U_L K}{l} \frac{\partial \tau_{imb}}{\partial U_L} + \frac{E_L K}{l} \frac{\partial(\tau_{idf} - \tau_{imb})}{\partial E_L} = 0 \quad (4.8)$$

Since both eigenvalues must be negative to obtain a stable solution, $\lambda_1 < 0$ and $\lambda_2 < 0$, i.e., $\lambda_1 \lambda_2 > 0$, or

$$\frac{U_L K}{l} \frac{\partial \tau_{imb}}{\partial U_L} + \frac{E_L K}{l} \frac{\partial(\tau_{idf} - \tau_{imb})}{\partial E_L} > 0 \quad (4.9)$$

Considering that $\tau_{imb} = f[U_{SL}(U_L, E_L), E_L]$ one can get

$$\left. \frac{\partial \tau_{imb}}{\partial E_L} \right|_{U_L} = \left. \frac{\partial \tau_{imb}}{\partial E_L} \right|_{U_{SL}} + \left. \frac{\partial \tau_{imb}}{\partial U_{SL}} \right|_{E_L} \left. \frac{\partial U_{SL}}{\partial E_L} \right|_{U_L} \quad (4.10)$$

Substituting Eq. 4.10 into Eq. 4.9 results in

$$\frac{\partial \tau_{idf}}{\partial E_L} > \frac{\partial \tau_{imb}}{\partial E_L} \quad i.e. \quad \frac{\partial \tau_{idf}}{\partial (h_L/D)} > \frac{\partial \tau_{imb}}{\partial (h_L/D)} \quad (4.11)$$

Eq. 4.11 implies that the slope of $\tau_{idf}-E_L$ curve or $\tau_{idf}-h_L/D$ curve should be greater than that of $\tau_{imb}-E_L$ curve or $\tau_{imb}-h_L/D$ curve. From Figs. 4.19, 4.20, 4.22, and 4.21 it is easily observed that only at the lowest and highest roots of liquid height and pressure gradient is Eq. 4.11 satisfied. For example, if we denote three interception points in Fig. 4.20 for $U_{SL} = 0.001 ft/sec$ as point *A*, point *B*, and point *C*, then at point *A* and point *C*, slopes of $\tau_{imb}-h_L/D$ curves are negative, while the slopes of $\tau_{idf}-h_L/D$ are positive. Hence Eq. 4.11 is satisfied at both points. At point *B*, though slopes of both $\tau_{idf}-h_L/D$ and $\tau_{imb}-h_L/D$ curves are positive, the slope of $\tau_{idf}-h_L/D$ curve is smaller than that of $\tau_{imb}-h_L/D$ curve, therefore, Eq. 4.11 can not be satisfied. It thus can be concluded that the *intermediate solution is unstable*.

Another method to check if the solution for liquid holdup and pressure gradient is realistic is to study the asymptotic behavior at small X . When the Lockhart-Martinelli parameter X goes to zero, unless gas flow rate is very low and thus inclination parameter $-Y$ is very large (note that Y is negative for upward flow), liquid flow rate becomes trivial, gas-liquid two-phase flow transforms to single phase gas flow, as a consequence, liquid height and liquid holdup should be zero, and pressure gradient approaches the pressure gradient for single phase gas pipe flow with the same gas flow rate. This trend is not observed for the highest root (point *C* in Fig. 4.20) and the intermediate root (point *B* in Fig. 4.20) cases, hence it is also concluded that *the highest root is physically not realistic*.

Another approach to reach same conclusion is to numerically solve the momentum balance equations for transient gas-liquid stratified flow. Details will be given in Section 4.8.

Based on the above observations, the following conclusion is reached: *only the lowest root of liquid height and liquid holdup (point A in Fig. 4.20) satisfies the stability condition of Eq. 4.11 and makes physical sense, hence, it should be the liquid height and liquid holdup to be encountered in reality*. Similar observations have been reported by Landman [2, 3] and Barnea & Taitel [4] for gas-liquid stratified flow in pipes where no wall mass transfer happens. However, Landman [2, 3] stated that both the lowest and the intermediate solutions are physically possible.

Finally it is worth mentioning that some kind of discontinuity in solution may occur when we go from the multiple solution region (*A* to *C* in Fig. 4.3) to the single solution region (*D* to *F*) if either the lowest or the intermediate root of liquid height is selected (the correct root to pick is the lowest one as shown above). In Fig. 4.3, liquid height will change

along the path: $A-B-C-D-E-F$. Inevitably, a jump or discontinuity in liquid height and thus pressure gradient will occur from C to D .

4.8 Numerical Validation of Multiple Solutions

Eqs. 4.5 and 4.7 describe an initial-value problems for liquid holdup and average liquid velocity. A number of discrete variable methods have been proposed for the solution of initial value problems like this. These methods can be classified into either one-step methods or multi-step methods. A one-step method uses only information obtained in the range of a single step from t_i to t_{i+1} . One typical and perhaps the most-popular one-step method is known as the Runge-Kutta method, which can further be divided into 2nd-order, 3rd-order, 4th-order or 5th-order Runge-Kutta formula. Experience has shown that the 4th-order Runge-Kutta formula is the most powerful method for solving initial-value problems and thus it will be applied here to simulate the transient evolution of the liquid holdup and liquid velocity.

For an initial-value system,

$$\frac{dy_i(t)}{dt} = f_i(t, y_1, y_2, \dots, y_N) \quad i = 1, 2, \dots, N \quad (4.12)$$

the 4th-order Runge-Kutta formula can be expressed as (Press *et al* [14])

$$\begin{cases} \vec{k}_1 = h\vec{f}(t_n, \vec{y}_n) \\ \vec{k}_2 = h\vec{f}(t_n + \frac{1}{2}h, \vec{y}_n + \frac{1}{2}\vec{k}_1) \\ \vec{k}_3 = h\vec{f}(t_n + \frac{1}{2}h, \vec{y}_n + \frac{1}{2}\vec{k}_2) \\ \vec{k}_4 = h\vec{f}(t_n + h, \vec{y}_n + \vec{k}_3) \end{cases} \quad (4.13)$$

and

$$\vec{y}_{n+1} = \vec{y}_n + \frac{1}{6}\vec{k}_1 + \frac{1}{3}\vec{k}_2 + \frac{1}{3}\vec{k}_3 + \frac{1}{6}\vec{k}_4 + O(h^5) \quad (4.14)$$

where the two vectors $\vec{y}$ and $\vec{f}$ are defined as

$$\vec{y} = (y_1, y_2, \dots, y_N)^T \quad \vec{f} = (f_1, f_2, \dots, f_N)^T$$

The above 4th-order Runge-Kutta method has been applied to solve the initial value problem composed of Eq. 4.5 and Eq. 4.7. Some of the simulation results are shown in Figs. 4.23 – 4.32 and will be discussed. Fig. 4.23 displays the evolution of liquid holdup and average liquid velocity with time. Starting with any reasonable initial state of liquid

holdup and liquid velocity (say the point marked by the blank triangle in Fig. 4.23), gas-liquid flow changes quickly with time, and soon it reaches a stable state. It is interesting to see that this state corresponds to the stabilized flow determined by the momentum balance equations described in Ouyang [12]. Based on the liquid holdup and average liquid velocity variation with time (Figs. 4.24 and 4.25), it is found that it takes a fairly short time for the system to reach the stabilized state or steady-state. Once the steady state is reached, no further change in liquid holdup and liquid velocity is expected, the time-dependent terms vanish, and transient flow equations described in Ouyang [12] are transferred into steady state equations. During the evolution from initial state to the final steady state, liquid holdup and liquid velocity change substantially (see Figs. 4.23). But the liquid velocity remains positive and never changes its sign. That means that liquid flow and gas flow are cocurrent all the time. This is a necessary condition to maintain stratified flow.

Different initial states of the gas-liquid system have been tried and it was observed that all transient states of the system become stabilized after a short time. A unique steady state was obtained for the system regardless of the initial conditions.

Figure 4.23: Flow Evolution Expressed in E_L-U_L Plane for Single Root Case ($U_{SG} = 60.0 \text{ ft/sec}$, $U_{SL} = 0.1 \text{ ft/sec}$, $\theta = 1^\circ$)

The kind of simple transient features discussed above does not exist for all the gas and liquid flow rates. For certain flow rates under which multiple solutions are anticipated, three instead of one steady state may be expected for the system. Figs. 4.26, 4.27 and 4.28

Figure 4.24: Liquid Holdup Change with Time ($U_{SG} = 60.0 \text{ ft/sec}$, $U_{SL} = 0.1 \text{ ft/sec}$, $\theta = 1^\circ$)

Figure 4.25: Average Liquid Velocity Change with Time ($U_{SG} = 60.0 \text{ ft/sec}$, $U_{SL} = 0.1 \text{ ft/sec}$, $\theta = 1^\circ$)

show the flow evolution for a system with $U_{SG} = 25.0 \text{ ft/sec}$, $U_{SL} = 0.005 \text{ ft/sec}$, and $\theta = 1^\circ$. Three steady state solutions exist for this system. Simulations with following five different initial states for the system have been conducted:

- A. $h_{LD} = 0.06$, $U_L = 0.6 \text{ ft/sec}$
- B. $h_{LD} = 0.11$, $U_L = 0.1 \text{ ft/sec}$
- C. $h_{LD} = 0.12$, $U_L = 0.1 \text{ ft/sec}$
- D. $h_{LD} = 0.50$, $U_L = 0.1 \text{ ft/sec}$
- E. $h_{LD} = 0.10$, $U_L = 0.3 \text{ ft/sec}$

For comparison purpose, the three steady states are also marked in the figures, where the solid rectangle represents the lowest solution of liquid holdup, the solid triangle the intermediate solution, and the solid circle the highest solution. Figs. 4.27 and 4.28 show a higher range of liquid holdups in which the highest solution (solid circle) is hidden under the solution path near the center of the figures. As expected, the system with initial condition near the lowest root evolves and reaches the lowest solution very quickly (Fig. 4.26). The evolution process for this type of initial condition is quite similar to that of the single solution case shown in Fig. 4.23. It is interesting to note that a system with an initial state close to the intermediate solution can have completely different final state depending upon where the initial state lies relative to the intermediate solution. Two different but close initial states were used to demonstrate the transient evolution of the gas-liquid stratified flow, one initial state ($h_{LD0} = 0.11$, $U_{LD0} = 0.1 \text{ ft/sec}$) is on the left side of the intermediate steady state location, while the other ($h_{LD0} = 0.12$, $U_{LD0} = 0.1 \text{ ft/sec}$) on the right side. Both initial states are really close to the intermediate state ($h_{LD} = 0.113$, $U_{LD} = 0.084 \text{ ft/sec}$). Surprisingly, the gas liquid flow does not develop into the intermediate solution with time increasing, instead, it reaches the lowest or the highest solutions (Figs. 4.26 and 4.27). The time to reach steady-state for the first case is short (Figs. 4.29 and 4.31), while it is long for the latter case which ends up at the highest solution (Figs. 4.30 and 4.32). Note that the average liquid velocity is always positive for the initial state of $h_{LD0} = 0.11$, $U_{LD0} = 0.1 \text{ ft/sec}$, which means that there is no back flow of the liquid phase. On the other hand, the liquid velocity becomes negative for certain time periods for the initial state of $h_{LD0} = 0.12$, $U_{LD0} = 0.1 \text{ ft/sec}$, which implies that liquid back flow occurs and that the stratified flow status can not be maintained. Therefore, the steady state stratified flow for the intermediate solution is unstable and can not be reached, which is consistent with the analysis conducted in Section 4.7 (Eq. 4.11).

The last initial state tested is $h_{LD0} = 0.5$, $U_{LD0} = 0.1 \text{ ft/sec}$, where the liquid height is much higher than all the three solutions for the steady state momentum balance equations. Again similar features to those for the initial state of $h_{LD0} = 0.12$, $U_{LD0} = 0.1 \text{ ft/sec}$ were observed (Figs. 4.28, 4.30 and 4.32).

Based on above simulation, it can be concluded that for initial states which have higher liquid holdup or liquid height than those for the intermediate solution, the gas-liquid stratified flow status may be broken and flow pattern change will occur before reaching a steady state corresponding to either intermediate or the highest solutions. Hence, once again it is shown that only the lowest solution is physically reasonable.

Figure 4.26: Flow Evolution Expressed in E_L - U_L Plane for Multiple Root Case (Initial State A and Initial State B)

4.9 Multiple Solutions for Annular-Mist Flow

Similar to stratified flow, there may also exist solution nonuniqueness for liquid film thickness for annular-mist flow. Due to high flow rate of gas phase, some liquid can be entrained in the gas core. Amount of liquid entrainment is dependent on many parameters, such as superficial gas and liquid velocities, among others. Density of gas core is thus a function of both superficial gas and liquid velocities. Hence it is difficult to obtain a constant inclination parameter Y by simply fixing the superficial gas velocity. Alternatively,

Figure 4.27: Flow Evolution Expressed in E_L - U_L Plane for Multiple Root Case (Initial State C)

Figure 4.28: Flow Evolution Expressed in E_L - U_L Plane for Multiple Root Case (Initial State D)

Figure 4.29: Liquid Holdup Change with Time - Multiple Root Case (Initial State *A* and Initial State *E*)

Figure 4.30: Liquid Holdup Change with Time - Multiple Root Case (Initial State *C* and Initial State *D*)

Figure 4.31: Average Liquid Velocity Change with Time - Multiple Root Case (Initial State *A* and Initial State *E*)

Figure 4.32: Average Liquid Velocity Change with Time - Multiple Root Case (Initial State *C* and Initial State *D*)

as suggested by Petalas & Aziz [5], dependence of liquid film thickness on superficial liquid velocity for different superficial gas velocities may be investigated. Results shown in Fig. 4.33 indicate that liquid entrainment has a significant effect on liquid film thickness. For certain gas flow rates such as $U_{SG} = 50 \text{ ft/sec}$ and $U_{SG} = 80 \text{ ft/sec}$, liquid film thickness does not continue to increase with increasing liquid flow rate as it would if no liquid entrainment occurred (see Fig. 4.34). Instead, liquid film thickness starts to decrease at certain liquid flow rate and then increases again after reaching some higher liquid flow rate. It is expected that decreasing of liquid film thickness is caused by liquid entrainment in the gas core, which reduces the liquid film thickness. Fig. 4.33 also shows that multiple solutions for liquid film thickness happen for $U_{SG} = 50 \text{ ft/sec}$ and $U_{SG} = 80 \text{ ft/sec}$. It is interesting to note that the features of multiple solutions are quite different for $U_{SG} = 50 \text{ ft/sec}$ and $U_{SG} = 80 \text{ ft/sec}$. The liquid film thickness curve in the single solution region connects to the highest root for $U_{SG} = 50 \text{ ft/sec}$, while it connects to the lowest root for $U_{SG} = 80 \text{ ft/sec}$. As we have found for stratified flow, only the lowest root of liquid film thickness is physically reasonable. Whenever multiple solutions display feature like that for $U_{SG} = 50 \text{ ft/sec}$, discontinuity is anticipated (Fig. 4.35). The curve for liquid film thickness selected will have a path like $A-C-D-E$. A jump from C to D can not be avoided.

Figure 4.33: Solution Nonuniqueness for Liquid Film Thickness in Annular-Mist Flow (No Wall Inflow/Outflow, $\theta = 10^\circ$)

As we stated in Section 4.4, it is better to express multiple solution regions by means

Figure 4.34: Solution Nonuniqueness for Liquid Film Thickness in Annular-Mist Flow (No Liquid Entrainment)

Figure 4.35: Discontinuity in Liquid Film Thickness Solution (No Wall Inflow/Outflow, $U_{SG} = 50.0$ ft/sec, $\theta = 10^\circ$)

of X - Y maps. Such a map is shown in Fig. 4.36 where two different interfacial friction factor correlations are applied, one is the Petalas & Aziz (1996) [15] correlation, and the other is $f_i = 0.05$. Surprisingly, substantial differences in solution nonuniqueness exist for the two different interfacial friction factor correlations. For the Petalas & Aziz (1996) correlation, multiple solutions exist in about a range of $Y = -50 \sim -10$. For the $f_i = 0.05$ correlation, the range becomes $Y = -600 \sim -70$. Based on the first interfacial friction factor correlation, the maximum Lockhart-Martinelli parameter for multiple solution to occur is around 0.3, while the value changes to 1.0 for the second correlation. Nevertheless, shape of the multiple solution region is quite similar, which implies that f_i correlation only quantitatively affects the multiple solution region but does not qualitatively alter its shape.

Since Fig. 4.36 is expressed in terms of two dimensionless variables, it is useful to wonder whether it can be applied for different fluid system, pipe geometry and flow rates. It is already shown that the X - Y map is indeed applicable for different flow system for stratified flow (Section 4.4). Unfortunately, this is not true for annular-mist flow. If pipe internal diameter is changed from 2.047 inch to 6.18 inch or if pipe inclination angle is set as 1° instead of 10° in Fig. 4.36, although the shape of envelope for multiple solution region does not change much, the region varies significantly (see Fig. 4.37 and Fig. 4.38). Part of the reason for this could be due to liquid entrainment in gas core.

Figure 4.36: Multiple Root Region in a X - Y Map

Figure 4.37: Multiple Root Region in a X - Y Map (Changing Inclination Angle)

Figure 4.38: Multiple Root Region in a X - Y Map (Changing Pipe Diameter)

Fig. 4.39 shows the multiple solution regions for the liquid inflow dominant case ($I = -10$) and the gas inflow dominant case ($I = 10$). Shapes of these regions do not change significantly. But the region shifts to the left for gas inflow dominant case while it shifts to the right for liquid inflow dominant case. Amount of shift in multiple solutions region is proportional to the absolute value of inflow parameter I . Therefore, for the liquid inflow dominant situation, if the inflow parameter is high enough, it is expected that region of solution nonuniqueness will extrude into the half plane $Y > 0$, which means that multiple solution may also occur for horizontal and downward gas-liquid annular-mist flow.

Figure 4.39: Influence of Wall Mass Transfer on the Multiple Root Region in a X - Y Map

4.10 Concluding Remarks

Occurrence of multiple solution has been investigated and discussed in this chapter. Multiple solution region is displayed by either a U_{SL} - U_{SG} map or a X - Y map. Physical consideration and numerical simulation of the transient process for separated flow have been conducted. The study has led to the following observations and conclusions:

- Multiple solutions may occur for gas-liquid two-phase separated in a pipe or a wellbore. This is only possible for upward flow for separated flow in a pipe without wall mass transfer. However, for the wellbore flow case, depending on the wall inflow/outflow of

fluids and their flow rates, multiple solution may also exist for horizontal and downward flow. More specifically, for wall inflow (production well) case, multiple solutions only exist for upward flow for the gas inflow-dominant case, whereas they may exist for upward, horizontal and downward flow for the liquid inflow-dominant situations. The opposite is expected for wall outflow (injection well) cases

- The multiple solution region depends on the interfacial friction factor correlation used in the stratified or annular flow model. The region could change significantly if different correlations are applied.
- Physical consideration and numerical simulations of a transient process have shown that only the solution with the lowest liquid height (holdup) is physically realistic. The solutions corresponding to the intermediate and the highest liquid holdup will not be lead to stable flow.

References

- [1] Baker, A. and Gravestock, N.: "New Correlations for Predicting Pressure Loss and Holdup in Gas/Condensate Pipelines," in *Proc., 3rd International Conference on Multiphase Flow*, The Hague, The Netherlands, 417-435, 1987.
- [2] Landman, M. J.: "Non-unique Holdup and Pressure Drop in Two-phase Stratified Inclined Pipe Flow," *Int. J. of Multiphase Flow*, 17, No. 3, 377-394, 1991.
- [3] Landman, M. J.: "Hysteresis of Holdup and Pressure Drop in Simulations of Two-phase Stratified Inclined Pipe Flow," in *Multiphase Production (edited by A. P. Burns), Proc. of the 5th International Conference on Multi-phase Production*, Cannes, France, June 19-21, 1991.
- [4] Barnea, D. and Taitel, Y.: "Structural and Interfacial Stability of Multiple Solutions for Stratified Flow," *Int. J. of Multiphase Flow*, 18, No. 6, 821-830, 1992.
- [5] Petalas, N. and Aziz, K.: "A Mechanistic Model for Stabilized Multiphase Flow in Pipes," Technical Report, Stanford University, 1997.
- [6] Ouyang, L-B.: *Stratified Flow Model and Interfacial Friction Factor Correlations* M.S. Report, Stanford University, June 1995.

- [7] Duns, Jr. H. and Ros, N. C. J.: "Vertical Flow of Gas and Liquid Mixtures in Wells," in *Proc. of The Sixth World Petroleum Congress*, Frankfurt, Germany, paper No. 22, 1963.
- [8] Baker, A., Nielsen, K. and Gabb, A.: "Pressure Loss, Liquid Holdup Calculations Developed," *Oil & Gas J*, **86** No. 11, 55-59, 1988.
- [9] Taitel, Y. and Dukler, A. E.: "A Model for Prediction of Flow Regime Transition in Horizontal and Near Horizontal Gas-Liquid Flow," *AIChE J*, **22**, No. 1, 47-55, 1976.
- [10] Cohen, L. S. and Hanratty, T. J.: "Generation of Waves in the Concurrent Flows of Air and a Liquid," *AIChE J*, **11**, No. 1, 138-144, 1965.
- [11] Andritsos, N. and Hanratty, T. J.: "Influence of Interfacial Waves in Stratified Gas-Liquid Flows," *AIChE J*, **33**, No. 3, 444-454, 1987.
- [12] Ouyang, L-B.: *Single Phase and Multiphase Fluid Flow in Horizontal Wells* PhD Dissertation, Stanford University, 1998.
- [13] Barnea, D. and Taitel, Y.: "Transient-Formulation Modes and Stability of Steady-State Annular Flow," *Chemical Engineering Science*, **44**, No. 2, 325-332, 1989.
- [14] Press, W. H., Teukolsky, S. A., Vetterling, W. T., and Flannery, B.: *Numerical Recipes in Fortran: The Art of Scientific Computing*, Cambridge University Press, England, 2nd Edition, 1992.
- [15] Petalas, N. and Aziz, K.: "Development and Testing of a New Mechanistic Model for Multiphase Flow in Pipes," in *2nd ASME International Symposium on Numerical Methods for Multiphase Flow*, San Diego, CA, July 7-11, 1996.

5. An Experimental Study of Single Phase and Two-Phase Flow in Horizontal Wells

by Liang-Biao Ouyang, Nicholas Petalas, Sepehr Arbabi, Donald E. Schroeder and Khalid Aziz*

Abstract

A brief overview of the 5-year Stanford horizontal wellbore experimental program is presented. Different existing mechanistic flow models, simplified flow models as well as empirical correlations are applied to analyze the 1995, 1996 and 1997 Stanford horizontal wellbore experiments. Comparisons between measurement and prediction are performed, which lead to the conclusion that none of the existing models can predict, with sufficient accuracy, pressure drops for gas-liquid wellbore flow with radial influx.

5.1 Introduction

Over the last decade horizontal wells have become a well-established technology for the recovery of oil and gas. They have become increasingly attractive for production from thin reservoirs, naturally fractured reservoirs, reservoirs with gas and water coning problems, offshore environments where various wells are drilled from a central platform, as well as in various enhanced oil recovery projects.

Modeling of horizontal wells poses additional challenges, one of which is the determination of the frictional pressure drop along the wellbore. The pressure drop due to friction over the pay zone is usually neglected in classical reservoir simulation models, because it is typically not an important factor in vertical wells. However, practical examples have shown that in many cases the inclusion of wellbore friction is essential for the correct prediction of well performance. For horizontal wells, the frictional pressure drop becomes even more significant since the drilling technology now allows wells to be several thousands of feet long. Hence, the frictional pressure drop over the production portion of the well will be much larger in horizontal wells compared to that in vertical wells. Furthermore, although the pressure drop due to friction in vertical wells is small compared to the change in pressure due to gravity, this is not the case in typical horizontal wells.

Depending on the completion method used in a horizontal well, fluid may enter the wellbore radially through perforations at various locations. This alters the flow behavior along the wellbore and complicates the modeling of the problem. For single-phase flow, Ouyang *et al.*¹ have shown that influx through well perforations increases wall friction in the laminar flow

* This paper (SPE 46221) was presented at the 1998 SPE Western Regional Meeting held in Bakersfield, California, 10–13 May 1998.

regime and decreases wall friction in the turbulent flow regime. This means that the frictional pressure drop can be larger or smaller than that for the case with no radial influx, depending on the flow regime present in the wellbore. In addition, as the distance between adjacent perforations may not be large enough to achieve a stabilized velocity profile, the flow characteristics and the pressure gradient in a horizontal wellbore may change with location. The flow behavior in a horizontal wellbore with increasing flow rate toward the heel and the relationship between pressure drop and influx from the reservoir are some of the most important but unsolved issues in the engineering analysis of such wells. Furthermore, the presence of multiple phases in the wellbore, already complicated through the existence of different flow patterns, is even further complicated by the presence of inflow.

To investigate the influence of inflow on single-phase and multiphase wellbore flow, a series of experiments were designed and conducted during the period, 1993 through 1997. The experiments were carried out in a 100-*ft* long horizontal wellbore model at the Marathon Oil Petroleum Technology Center in Littleton, Colorado. More than 1000 experimental runs featuring different combinations of axial flow and radial influx were performed during this 5-year program. Different wellbore IDs, wellbore types, gravel packs, perforation types and densities, fluid types and wellbore inclination angles were considered. Oil/nitrogen and air/water were used in the 1993 and 1994 experiments, while air and water were used in the 1995, 1996 and 1997 experiments. For 1996 and 1997, the wellbore model was modified by inserting stainless steel wire-wrapped screens inside the base acrylic pipe. Air-water flow experiments, similar to the 1995 experiments, were carried out using this setup (see next section).

The Stanford Horizontal Wellbore experiments are briefly introduced in the present paper. Different existing mechanistic models, simplified models as well as empirical correlations are applied to analyze the 1995, 1996 and 1997 experiments. The results of comparisons between measurement and prediction are also presented.

5.2 Experimental Setup and Types of Experiments

For each series of experiments, different wellbore setups have been applied. **Figure 5.1a** shows a simple sketch of the wellbore model used for the 1993 and 1994 experiments. The wellbore is essentially a transparent acrylic pipe with an average internal diameter of 6.2 *inch*. The layout consists of 20 *ft* of blank acrylic, followed by 20 *ft* of smooth perforations and 15 *ft* sharp perforations. Liquid or gas inflow can be supplied by the manifold connected to the next 40 *ft* of smooth edged perforations. The remaining 5 *ft* of the model is a blank section of the acrylic casing. There are small variations in the diameter of the wellbore which have been measured by mechanical and ultrasonic techniques.

The setup for the 1995 experiments is shown in **Figure 5.1b**. The wellbore test section is 100-*ft* long and consists of twenty 5-*ft* sections. The perforations begin at the 85 *ft* location and end at the 10 *ft* location. The wellbore internal diameter ranges from 6.15 *inch* to approximately 6.30 *inch*. This range is smaller than that of the wellbore employed for the 1994 experiments. Except

for the pipe sections at 20-25 *ft*, 55-60 *ft* and 80-95 *ft*, variation in the pipe internal diameter is quite small (less than 0.05 *inch*).

The model for 1996 experiments is shown in **Figure 5.1c**. The outer acrylic pipe is the same as that used during the 1995 experiments. A photograph of this model is shown in **Figure 5.2** - it is 100-*ft* long and consists of twenty 5-*ft* sections with an average internal diameter of 6.2 inches. The inner pipe contains three acrylic window sections (denoted by W in **Figure 5.1c**), each about 4-*ft* long, and three stainless steel pre-packed wire-wrapped screen sections, each about 21-*ft* long. These can be seen in **Figure 5.3**, showing a close-up photograph of a test section. Packers are used to eliminate flow in the annular space around the window sections. Inflow only occurs in the screen sections so that the total inflow length is approximately 63 *ft*; less than the 75 *ft* used in the 1995 experiments. Window sections are used to observe and record the flow regimes and they each house a nuclear densitometer to measure the average in situ liquid fraction (see **Figure 5.3**). The screen sections have very small variations in diameter (4.500 to 4.509 *inch*), whereas the window sections show somewhat larger variations in their diameters (4.450 to 4.476 *inch*). However, both variations are much smaller than the variations in diameters of the sections forming the outer pipe.

Two sets of experiments were conducted in 1997 (Table 5.1). One set (1997-B in Table 5.1) focused on air-water flow in a wellbore as sketched in **Figure 5.1c** with a 40/60 mesh sand-packed annulus. The other set of experiments (1997-A in Table 5.1) with the wellbore model shown in **Figure 5.1b** was performed-with perforation density variations from 0.5 *shots/ft* to 8 *shots/ft*.

The more than 1000 experimental runs featuring different combinations of axial flow and inflow conducted in the five-year program, can be classified under the following categories:

- A. wellbore characterization experiments;
- B. single-phase liquid axial flow with liquid inflow;
- C. single-phase liquid axial flow with gas inflow;
- D. gas and liquid two-phase axial flow;
- E. gas and liquid two-phase axial flow with liquid inflow;
- F. gas and liquid two-phase axial flow with gas inflow;
- G. gas and liquid two-phase axial flow with gas and liquid inflow.

Detailed information regarding pipe inclination angle and ranges of gas and liquid flow rate for each type of experiment is listed in Table 5.2.

For each experimental run, the following data were measured and automatically recorded into a spreadsheet file for later analysis:

- axial flow and inflow flow rate
- liquid fractions at 11.25 *ft*, 36.25 *ft*, and 61.25 *ft* locations

- pressure drops every 10 *ft* and 50 *ft*
- pipe inclination angle
- inlet and outlet temperatures and pressures.

For most experiments, data were recorded for at least 10 *mins* after reaching a stabilized state. The flow along the entire wellbore was video-taped and the flow patterns at 11.25 *ft*, 36.25 *ft* and 61.25 *ft* locations were also recorded.

A significant number of characterization experiments were conducted each year prior to the ones scheduled. Furthermore, in order to ensure accurate and reliable experimental data, a number of the experiments were repeated (some of them being repetitions of experiments from the previous year) and measurements compared.

5.3 Analysis of Experimental Data

5.3.1 Wellbore Calibration Experiment

Intuitively, the effective wellbore roughness will be altered (increased) due to the perforations drilled in the wellbore. The determination of the effective wellbore pipe roughness is an important issue for data analysis.

Kloster² experimentally studied flow resistance in a perforated pipe without inflow through the perforations by using a 6-5/6 *inch* OD and 17-*ft* long pipe. The pipe had a 14-*ft* long perforated section with a perforation density of 180 *shots/ft*. The Reynolds number varied from 60,000 to 450,000. He found that the friction factor vs. Reynolds number relationship didn't exhibit the characteristics of regular pipe flow. The friction factor for the perforated pipe did not decrease with increasing Reynolds number over the Reynolds number range investigated. It reached a local maximum between $Re = 160,000$ and $Re = 220,000$, where the friction factor increased by about 60-70% over the value determined by Reynolds number and pipe roughness. For Reynolds numbers less than 160,000, the friction factor increased by about 55%, while it increased by about 25-45% for Reynolds numbers greater than 220,000. Su & Gudmundsson³ measured the mass flow rates and water column heights for single-phase water flow along a vertical tube. To make comparisons, both the experimental data for flow before and after perforation drilling were obtained. Based on their measurements, Su & Gudmundsson³ introduced a roughness function that was correlated as a function of the perforation/casing diameter ratio.

Although the analyses associated with the above two studies for single-phase water flow in a perforated pipe may need to be improved, the observations do imply that the effective pipe roughness is augmented by perforations.

In order to determine the effective pipe roughness for the wellbore used in our experiments, calibration experimental runs were performed each year before the actual runs. Data from these

calibration experiments were applied to determine the effective or equivalent pipe roughness. By changing the effective pipe roughness, the best match between predicted and measured pressure drops for each section was determined.

5.3.2 Single-phase Water Wellbore Flow

As expected, it is observed that the higher the axial water flow rate, or the higher the water inflow rate, the higher will be the pressure drop. A more revealing comparison is presented in **Figure 5.4**, where two single-phase water flow experiments with the same amount of total water flow rate of 17.5 *MbpD* are compared: one with only axial water flow, the other with 10.5 *MbpD* of axial flow and 7.0 *MbpD* of inflow. Along the first half of the wellbore, the pressure drops for the 10-ft sections with no inflow are higher than for the case with inflow, while along the second half of the wellbore, pressure drops for the case with no inflow are lower than for the case with inflow. Note that from the inlet (100 ft location) to the 10 ft wellbore location, the average local water velocity is always higher for the case with no inflow than for the case with inflow. Therefore, it is very natural to conclude from **Figure 5.4** that inflow increases the pressure drop along the wellbore. Some researchers, such as Shapiro *et al.*⁴, Kays & Crawford⁵, use an *apparent* friction factor to describe the pressure drop along the wellbore. The above observation is then equivalent to saying that inflow increases the apparent friction factor. The statement is correct but misleading, since it is easy for engineers to infer that inflow increases wall friction.

Figure 5.5.5 demonstrates why this inference is inappropriate. It shows the variation of three friction factors with distance along the wellbore:

- The total (apparent) friction factor obtained on the basis of measurements,
- the actual friction factor obtained from measurement and a new single-phase wellbore flow model¹, and
- the no-wall-flow friction factor obtained from the Colebrook-White⁶ equation by using the local Reynolds number and the effective pipe roughness.

The apparent friction factor is larger than the no-wall-flow friction factor, but the actual friction factor is smaller than the no-wall-flow friction factor. In other words, the wall friction factor is reduced (not augmented) by inflow. This can be easily explained by the inflow and outflow influence mechanisms detailed in Ouyang *et al.*⁸, and it is a result of the turbulent flow regime present in all the single-phase water flow experiments.

5.3.3 Two Phase Wellbore Flow

Observations based on experimental data are summarized in this section.

Air inflow rate effect. The higher the air inflow rate, the higher the pressure drop, provided the

flow pattern doesn't change significantly. It is interesting to compare two wellbore flows with the same total air and water flow rates, one with 1.7 *MMscfD* air axial flow and 14.0 *MbpD* water axial flow, and the other with 14.0 *MbpD* water axial flow and 1.7 *MMscfD* air inflow. The overall pressure drop in the first case where no inflow is present is lower than that for the latter case with inflow. The main reason for this difference is due to the inflow effect on the accelerational and frictional pressure drops, since flow patterns observed for both cases are the same, i.e., slug flow along most of the wellbore.

Two-phase inflow effect. The inflow not only affects the accelerational and frictional pressure drops for two-phase wellbore flow, it may also modify the flow pattern present in the wellbore. As a result the pressure drop may not increase with air or water inflow. An example of this situation is given in **Figure 5.6**. The overall wellbore pressure drop for the case without inflow is even higher than for the case with inflow. This observation is different from what was observed for single-phase flow (**Figure 5.4**). Thus, flow pattern changes play an important role in determining the pressure drop along the wellbore during two-phase flow.

Inflow pattern effect. The air and water inflow distribution along the perforated part of the wellbore is referred to in this paper as the *inflow pattern*. Three inflow patterns were implemented in the 1995 experiments,

- Pattern #1: Air inflow in the first perforated half of wellbore (47.5-85 *ft*), water inflow in the second half (10-47.5 *ft*);
- Pattern #2: Water inflow in the first perforated half of wellbore (47.5-85 *ft*), air inflow in the second half (10-47.5 *ft*);
- Pattern #3: Alternating air and water inflow along the entire perforated wellbore.

Figure 5.7 and **Figure 5.8** show the influence of inflow pattern on the pressure drop for both low and high flow rate cases. For the low flow rate case, the overall pressure drop along the wellbore for inflow pattern #2 is 0.8 *psi* (Table 5.), which is about twice that for inflow pattern #1 and inflow pattern #3. Such a significant difference is related to changes in flow pattern. Even when the air and water axial and inflow rates are the same, the flow patterns observed are different for different inflow patterns. Stratified wavy flow occurs along the entire wellbore for both inflow patterns #1 and #3, whereas slug flow is observed along a part of the wellbore for inflow pattern #2. For the high flow rate case, slug flow occurs for all three inflow patterns and, as a consequence of this, the overall pressure drops along the wellbore are approximately the same (Table 5., **Figure 5.8**).

Brief summary. As discussed above, many factors come into play with two-phase flow in the wellbore, among them are axial air and water flow rates, air and water inflow rates, inflow pattern, pipe inclination, fluid properties, pipe geometry as well as perforation density, and perforation geometry. **Figure 5.9** compares pressure drops for different air and water flow arrangements in the wellbore where the total air and water flow rates are 1.0 *MMscfD* and 7.0 *MbpD*, respectively. The overall pressure drops along the wellbore are listed in descending order

in Table 5.. Both **Figure 5.9** and Table 5. show that pressure drops change significantly with flow arrangement. The overall pressure drop ranges from 0.2992 *psi* to 0.8891 *psi*. With air and water flow rates of 1.0 *MMscfD* and 7.0 *MbpD*, respectively, the flow format corresponding to water axial flow with air inflow (type C) and that corresponding to air and water axial flow with air inflow (type F) give the highest overall pressure drop. The lowest overall pressure drop is obtained with air and water axial flow with no inflow (type D).

Finally, it should be noted that influences of different parameters on wellbore flow behavior are interrelated and depend upon specific flow conditions. For example, the effect of the two-phase inflow pattern is dependent on the flow rates. For the low flow rate case, the overall pressure drop along the wellbore for inflow pattern #2 is about two times that for inflow patterns #1 and #3 (Table 5.4). The overall pressure drops, however, are approximately the same for all three inflow patterns when the flow rates are high (Table 5.4).

5.4 Prediction by Existing Models

In order to evaluate the predictive abilities of existing models and their application to two-phase flow with inflow, the following methods were selected:

- Petalas & Aziz⁹ mechanistic model,
- Xiao *et al.*¹⁰ mechanistic model,
- Beggs & Brill¹¹ model,
- Dukler¹² *et al.*¹² model,
- Homogeneous model.

The first three models require the determination of the flow pattern in order to calculate the pressure drop and holdup. Although the mechanistic approach attempts to account for more of the physical properties of the system, all of the above models rely on the use of one or more empirically determined correlations. One of these empirical relationships involves the determination of the single-phase friction factor. The traditional approach requires the use of a pipe roughness coefficient and a Reynolds number.

5.4.1 Pipe Roughness Determination

In order to determine an appropriate pipe roughness to use for the analysis of each of the 1995 and 1996 two-phase experimental data, the single-phase results from each year's experiments were used, with and without inflow. A pipe roughness was selected (1.0×10^{-4} *ft* for the 1995 experiments and 1.6×10^{-4} *ft* for 1996) such that the calculated pressure drop matched the experimental measurements. **Figure 5.10** shows a comparison between the experimentally measured pressure drop for the 1996 experiments compared with the value obtained by using traditional methods and ignoring the acceleration effects due to inflow. Although excellent agreement is obtained for the cases where no inflow is present, it is observed that, in the cases with inflow, neglecting the accelerational pressure losses yields extremely poor results.

By using a new wellbore flow model¹ that accounts for these acceleration losses due to inflow

as well as the effects of wall influx on friction, excellent agreement can be obtained between the calculated and the experimental values, as shown in **Figure 5.11**.

5.4.2 Analytical Procedure

The analysis of the two-phase experimental data was performed using the segmented model approach depicted in **Figure 5.12**. The procedure follows the steps outlined below:

- 1) The wellbore is divided into small segments; the smaller the segment, the more accurate the predictions.
- 2) Starting with the toe of the well, the calculations are performed by adding the air and water inflow rates for the segment, dQ_G and dQ_L , to the air and water axial flow rates, Q_G and Q_L .
- 3) The average superficial velocity for each phase is then computed.
- 4) Fluid properties are calculated at the average temperature and pressure in the segment (by assuming a pressure drop for the segment for the first iteration).
- 5) For models that require it, the predicted flow pattern for the segment is calculated from the superficial velocities and fluid properties.
- 6) The pressure drop and liquid holdup for the segment are then calculated as per the selected multiphase flow model.
- 7) If the calculated pressure drop exceeds a tolerance value (0.005 *psi*), the average segment pressure is recalculated based on the new pressure drop and steps 4) through 6) are repeated.
- 8) If the calculated pressure drop exceeds a maximum allowable value for the segment (0.1 *psi*), the segment is then divided into two segments and steps 4) through 7) are repeated.
- 9) After the pressure drop calculation has converged, the next segment is selected and steps 2) through 7) are repeated until the heel of the well is reached.
- 10) The pressure drop and liquid holdup values for all the segments along a section of interest are added and compared to the experimental measurements.

The above procedure was followed for each of the models listed above and the results thus obtained were compared to the experimental measurements for each of the years, 1995 and 1996, as well as some of the measurements from 1997.

5.4.3 Discussion of Results

The comparison with experimental data for all the multiphase flow models yielded extremely poor results. The results from the 1995 experiments displayed the worst behavior, chiefly due to the fact that the pressure drops were low in magnitude, thus making it difficult to obtain accurate measurements. Although the comparisons with the 1996 data showed an improvement over those for 1995, most of the models consistently over-predicted the pressure drop. This is apparent in **Figure 5.13** where the comparison based on the mechanistic model of Petalas & Aziz⁹ is displayed. It is seen that for the majority of the data, the calculated values exceeded the experimental observations by more than 25%. This behavior was consistent with all of the other models investigated, the Xiao *et al.*¹⁰ model failing to converge for a significant number of points. It is significant that, even for the cases where no inflow is present, the models still tend to greatly over-predict the pressure drop. Surprisingly, among the five models evaluated, the best match was obtained from the relatively simple homogeneous model. This is shown in **Figure 5.14**. However, it is observed that even here the pressure drop is poorly predicted when inflow is present. Also note that the accelerational pressure drop caused by inflow was not included in these models, as a consequence, the predicted pressure drop is expected to increase nontrivially for the cases with inflow.

There are a number of causes that can explain why the existing methods fail in providing good estimates when inflow exists in the wellbore:

1. All of the existing models are based on steady state, stabilized flow with no inflow. In two-phase wellbore flow with mass transfer through perforations, however, transitional flow conditions are observed even under stabilized flow conditions.
2. The large diameter pipes used in the experiments may require longer lengths in order to attain stabilized flow. This could account for the poor predictions by the models when there was no inflow present.
3. The flow pattern characteristics for pipe flow without inflow are significantly different than those for the same flow pattern when inflow occurs. Thus, the empirical correlations used in existing models, such as those used to determine Reynolds number, wall friction factor, interfacial friction factor, liquid entrainment, etc may be inappropriate. Furthermore, the transition mechanisms used in determining the flow pattern will likely differ when inflow occurs.
4. The calculation of holdup when inflow is present may differ significantly from the approach used by existing models. Overestimating this quantity may also lead to the kind of over-predictions of pressure drop observed here.
5. The pressure drop along a wellbore is composed of five components: frictional, gravitational, inflow-directional, accelerational due to inflow, and accelerational due to fluid

compressibility. Although the gravitational and inflow-directional components are trivial for horizontal wells¹, the accelerational component due to inflow can be large depending on the flow conditions. All the models or correlations evaluated here neither account for the accelerational pressure drop due to inflow nor for the inflow effect on the frictional pressure drop.

In order to evaluate a simple two-phase model that can account for the effects of inflow, a preliminary homogeneous model¹³ with slip between the phases is introduced. The model accounts for the effects of inflow by using an extension of the single-phase wellbore flow model presented by Ouyang *et al.*¹ This is presented in Figure 5.15 with results shown for the 1996 experiments. The results for most of the cases are excellent, most of them being well within a 25% error margin. The fact that this model is better able to predict the "no inflow" cases (where inflow accelerational effects do not exist) when compared to the no-slip homogeneous model (Figure 5.14), stems from introducing new definitions of the two-phase density and the two-phase velocity that honor mass balance in the system.

Although preliminary, simple models such as this show promise, especially since it is generally difficult under field conditions to determine the flow distribution, rates, etc of the inflowing fluids.

5.5 Development of New Wellbore Flow Models

One of the major objectives for conducting the 5-year horizontal well experiments is to obtain data to develop and validate new models or correlations. As mentioned above, a new single-phase wellbore flow model was developed in 1996¹. It was found that the new model provided excellent predictions for pressure drops for the 1995, 1996 and 1997 single-phase water experiments.

New two-phase wellbore flow models (Ouyang¹³), including a simplified homogeneous model and a mechanistic model, have been proposed and are currently being tested and improved.

5.6 Summary and Observation

The 5-year Stanford horizontal wellbore experimental program was introduced. A segmented approach has been applied to predict flow pattern, pressure drop and liquid holdup by using existing mechanistic models or empirical correlations. Comparisons between model predictions and measurements were made for the 1995, 1996 and 1997 experiments. Substantial differences between measurements and predictions have been observed. Some of the possible reasons are given below:

- Existing models or correlations do not account for the inflow effect, even though the kinetic energy effect due to inflow can be quite significant.
- Correlations for wall friction factors, interfacial friction factor and liquid entrainment fraction

were originally proposed for gas-liquid two-phase pipe flow without inflow, as a consequence they may not be appropriate for two-phase wellbore flow when inflow is present.

- Some of the transition criteria in existing mechanistic models may be inappropriate for wellbore flow with influx, hence the flow patterns predicted may be incorrect.
- New models or correlations need to be proposed for two-phase wellbore flow where radial inflow/outflow exists.

Nomenclature

WCQ	=	water axial flow rate
WIQ	=	water radial inflow rate
WQ	=	water flow rate, either inflow or axial flow rate
ACQ	=	air axial flow rate
AIQ	=	air radial inflow rate
AQ	=	air flow rate, either inflow or axial flow rate
WAQ	=	water and air flow rate, or, $WAQ = (WCQ, ACQ, WIQ, AIQ)$

Acknowledgements

The work was partly supported by the Stanford University Petroleum Research Institute's Reservoir Simulation Industrial Affiliates Program (SUPRI-B), the Stanford Project on the Productivity and Injectivity of Horizontal Wells (SUPRI-HW), and the DOE contract #DE-FG22-93BC14862.

References

1. Ouyang, L-B., Arbabi, S., and Aziz, K. "General Single Phase Wellbore Flow Model for Horizontal, Vertical and Slanted Well Completions," SPE paper 36608, presented at the 71st Annual SPE Technical Conference & Exhibition, Denver, Colorado, 1996
2. Kloster, J. "Experimental Research on Flow Resistance in Perforated Pipe," M. S. report, Norwegian Institute of Technology, 1990.
3. Su, Z., and Gudmundsson, J. S. "Friction Factor of Perforation Roughness in Pipes," paper SPE 26521, presented at the 68th SPE Annual Technical Conference and Exhibition, Houston, Texas, Oct 3-6, 1993
4. Shapiro, A. H., Siegel, R., and Kline, S. J. "Friction Factor in the Laminar Entry Region of a Smooth Tube," Proc. Second U. S. National Congress of Applied Mechanics, Ann Arbor, MI, June 14-18, 1954, 733-741.

5. Kays, W. M., and Crawford, M. E. *Convective Heat and Mass Transfer*, McGraw-Hill Co., New York, 1993, 601pp.
6. Colebrook, C. F. "Turbulent Flow in Pipes, with Particular Reference to the Transition Region between the Smooth and Rough Laws," *J. Inst. Civil Engineering* (London), vol 12, 133-156, 1939.
7. Colebrook, C. F., and White, C. M. "Experiments with Fluid Friction in Roughened Pipes," *Proc. Roy. Soc.* (London), vol 161A, 367-381, 1937.
8. Ouyang, L-B., Arbabi, S. and Aziz, K. "General Single Phase Wellbore Flow Model," US DOE Topical Report, 1997.
9. Petalas, N., and Aziz, K. "Development and Testing of a New Mechanistic Model for Multiphase Flow in Pipes," Second International Symposium on Numerical Methods for Multiphase Flows, ASME Fluids Engineering Division Summer Meeting, July 7-11, 1996, San Diego, California, USA.
10. Xiao, J. J., Shoham, O. and Brill, J. P. "A Comprehensive Mechanistic Model for Two-Phase Flow in Pipelines," SPE paper 20631, presented at the 65th SPE Annual Technical Conference & Exhibition, New Orleans, LA, 1990
11. Beggs, H. D., and Brill, J. P. "A Study of Two-Phase Flow in Inclined Pipes," *Trans. AIME*, vol 256, 607, 1973.
12. Dukler, A. E., Wicks, M., and Cleveland, R. G. "Frictional Pressure Drop in Two Phase Flow: A. A Comparison of Existing Correlations for Pressure Loss and Holdup," *AIChE J.*, vol 10, 38, 1964.
13. Ouyang, L-B. "Single Phase and Multiphase Fluid Flow in a Wellbore," Ph D Dissertation, Stanford University, Stanford, CA, 1998

Table 5.1. Summary of Wellbore Setup

Year	Wellbore ID (inch)	Hole ID (inch)	Screen ?	Sand Pack ?	Perf. Density
1993	6.2	6.2	no	no	uniform
1994	6.2	6.2	no	no	uniform
1995	6.2	6.2	no	no	uniform
1996	4.5	6.2	yes	no	uniform
1997-A	6.2	6.2	no	no	variable
1997-B	4.5	6.2	yes	yes	uniform

Table 5.2. Flow Rate Ranges

Type of Exp	Axial Liquid (MbpD)	Axial Gas (MMscfD)	Liquid Inflow (MbpD)	Gas Inflow (MMscfD)	Pipe Angle (degree)
1993 Experiments (oil/nitrogen, air/water)					
A	20	n/a	n/a	n/a	0
B	14--17	n/a	3.5	n/a	0
C	6.8--15	n/a	n/a	0.1--0.3	0
D	6.8--15	0.1--0.3	n/a	n/a	0
E	n/a	n/a	n/a	n/a	n/a
F	n/a	n/a	n/a	n/a	n/a
G	n/a	n/a	n/a	n/a	n/a
1994 Experiments (oil/nitrogen, air/water)					
A	20	n/a	n/a	n/a	0
B	14--17	n/a	3.5	n/a	0
C	6.8--15	n/a	n/a	0.1--0.3	0
D	6.8--15	0.1--0.3	n/a	n/a	0
E	n/a	n/a	n/a	n/a	n/a
F	n/a	n/a	n/a	n/a	n/a
G	n/a	n/a	n/a	n/a	n/a
1995 Experiments (air/water)					
A	14--30	n/a	n/a	n/a	0
B	7.0--18	n/a	3.5--11	n/a	-2, 0,+2
C	3.5--14	n/a	n/a	0.5--1.7	-2, 0,+2
D	3.5--14	0.5--1.7	n/a	n/a	-2, 0,+2
E	3.5--14	0.5--1.7	3.5	n/a	-2, 0,+2
F	3.5--14	0.5	n/a	0.5--1.0	-2, 0,+2
G	3.5--14	0.5	3.5	0.5--1.0	0
1996 Experiments (air/water)					
A	17--30	n/a	n/a	n/a	-2, 0,+2
B	7.0--18	n/a	1.0--11	n/a	0
C	3.5--14	n/a	n/a	0.5--1.7	-2, 0,+2
D	3.5--14	0.5--1.8	n/a	n/a	-2, 0,+2
E	3.5--14	0.5--1.8	3.5	n/a	-2, 0,+2
F	3.5--14	0.5	n/a	0.5--1.0	-2, 0,+2
G	3.5--14	0.5--1.0	3.5	0.5--1.0	0
1997 Experiments--Phase A (air/water)					
A	3.0--30	n/a	n/a	n/a	0
B	2.8--7	n/a	0.6--9	n/a	0
C	n/a	n/a	n/a	n/a	n/a
D	3.5--21	0.5--2.2	n/a	n/a	0,+2

Type of Exp	Axial Liquid (MbpD)	Axial Gas (MMscfD)	Liquid Inflow (MbpD)	Gas Inflow (MMscfD)	Pipe Angle (degree)
E	16.0	2.2	1.5--8	n/a	0,+2
F	n/a	n/a	n/a	n/a	n/a
G	n/a	n/a	n/a	n/a	n/a
1997 Experiments--Phase B (air/water)					
A	3.0--30	n/a	n/a	n/a	-2, 0,+2
B	10--14	n/a	10.3	n/a	-2, 0,+2
C	n/a	n/a	n/a	n/a	n/a
D	n/a	n/a	n/a	n/a	n/a
E	10--14	1.0--1.8	3.4	n/a	-2, 0,+2
F	n/a	n/a	n/a	n/a	n/a
G	n/a	n/a	n/a	n/a	n/a

Table 5.3. Overall Pressure Drop Comparison

Flow Format	Overall pressure drop (<i>psia</i>)
Water axial flow with air inflow	0.8891
Air and water axial flow with air inflow	0.8856
Air and water axial flow and inflow, pattern #2	0.8001
Air and water axial flow with water inflow	0.5575
Air and water axial flow and inflow, pattern #3	0.4108
Air and water axial flow and inflow, pattern #1	0.3763
Air and water axial flow	0.2993

Table 5.4. Inflow pattern effect at high flow rate

Flow Format	Overall pressure drop (<i>psia</i>)
Air and water axial flow and inflow, pattern #1	3.827
Air and water axial flow and inflow, pattern #3	3.507
Air and water axial flow and inflow, pattern #2	3.288

(a). 1993 and 1994 Wellbore Model

(b). 1995 Wellbore Model

(c). 1996 Wellbore Model

Figure 5.1. Wellbore test section used for the Stanford horizontal well experiments

Nuclear Liquid
Densitometer

Scanning Video
Camera

Radial Flow through
Perforations

Figure 5.2. Photograph of wellbore test section used for the 1996 Stanford horizontal well experiments

Wire-wrapped
Insert Screens

Start of Window
Section

Radial Flow through
Perforations (8 shots/ft)

Figure 5.3. Close-up photograph of wellbore test section used for the 1996 Stanford horizontal well experiments

Figure 5.4. Water inflow effect on pressure drop—comparison between two cases with same total water flow rate

Figure 5.5. Comparison among different fanning friction factors

Figure 5.6. Two-phase inflow rate effect—comparison between two-phase axial flow with and without two-phase inflow

Figure 5.7. Two-phase inflow pattern effect—low flow rate case, WAQ = (3.5, 0.5, 3.5, 0.5)

Figure 5.8. Two-phase inflow pattern effect—high flow rate case, WAQ = (14.0, 0.5, 3.5, 1.0)

Figure 5.9. Pressure drop comparison at 1.0MMscfD air and 7.0MbpsD water flow rates

Figure 5.10. Comparison between calculated and measured pressure drop for single-phase flow (Inflow acceleration effects ignored, 1996 experiments)

Figure 5.11. Comparison between calculated and measured pressure drop for single-phase flow (Inflow acceleration effects included, 1996 experiments)

Figure 5.12. Segmented model approach

Figure 5.13. Comparison of two-phase flow pressure drop between experimental measurements and values obtained from the Petalas and Aziz mechanistic model (Inflow acceleration effects ignored, 1996 experiments)

Figure 5.14. Comparison of two-phase flow pressure drop between experimental measurements and values obtained from a no-slip homogeneous model (Inflow acceleration effects ignored, 1996 experiments)

Figure 5.15. Comparison of two-phase flow pressure drop between experimental measurements and values obtained from Ouyang (1997) homogeneous model with slip (Inflow acceleration effects included, 1996 experiments)

6. A Mechanistic Model for Multiphase Flow in Pipes

*by Nicholas Petalas and Khalid Aziz**

Abstract

Mechanistic models for multiphase flow calculations can improve our ability to predict pressure drop and holdup in pipes especially in situations that cannot easily be modeled in a laboratory and for which reliable empirical correlations are not available. In this paper, a new mechanistic model, applicable to all pipe geometries and fluid properties is presented. New empirical correlations are proposed for liquid/wall and liquid/gas interfacial friction in stratified flow, for the liquid fraction entrained and the interfacial friction in annular-mist flow, and for the distribution coefficient used in the determination of holdup in intermittent flow.

6.1 Introduction

Empirical models often prove inadequate in that they are limited by the range of data on which they were based and, generally, cannot be used with confidence in all types of fluids and conditions encountered in oil and gas fields. Furthermore, many such models exhibit large discontinuities¹ at the flow pattern transitions and this can lead to convergence problems when these models are used for the simultaneous simulation of petroleum reservoirs and associated production facilities. Mechanistic models, on the other hand, are based on fundamental laws and thus can offer more accurate modeling of the geometric and fluid property variations.

All of the models presented in the literature are either incomplete^{2,3}, in that they only consider flow pattern determination, or are limited in their applicability to only some pipe inclinations^{4,5}. A preliminary version⁶ of the model proposed here that overcomes these limitations was presented in 1996.

For most of the flow patterns observed, one or more empirical closure relationships are required even when a mechanistic approach is used. Where correlations available in the literature are inadequate for use in such models, new correlations must be developed. In order to be able to achieve this, access to reliable experimental data is important.

* This paper (CIM 98-39) was presented at the 49th Annual Technical Meeting of the Petroleum Society of the Canadian Institute of Mining, Metallurgy and Petroleum held in Calgary, Alberta, Canada on June 8-10, 1998.

A large amount of experimental data has been collected through the use of a Multiphase Flow Database⁷ developed at Stanford University. The database presently contains over 20,000 laboratory measurements and approximately 1800 measurements from actual wells. Based on subsets of these data, the previously proposed model⁶ included a detailed investigation of the annular-mist flow regime and new correlations for the liquid fraction entrained and for interfacial friction. This model has since been refined based on additional investigations of the stratified and intermittent flow regimes, and is the subject of this paper.

6.2 Flow Pattern Determination

The procedure for flow pattern determination begins with the assumption that a particular flow pattern exists and is followed by an examination of various criteria that establish the stability of the flow regime. If the regime is shown to be unstable, a new flow pattern is assumed and the procedure is repeated. Figure 6.1 shows flow pattern transitions based on the superficial velocities of the phases where the stability criteria considered in this model are sketched. The procedure for flow pattern determination is illustrated in Figure 6.3, where it is seen that the examination of the dispersed bubble flow regime is the first to be considered.

6.2.1 Dispersed Bubble Flow

The dispersed bubble flow region is bounded by two criteria. The first is based on the transition to slug flow proposed by Barnea² where a transition from intermittent flow occurs when the liquid fraction in the slug is less than the value associated with the maximum volumetric packing density of the dispersed bubbles (0.52):

$$E_{Ls} < 0.48 \quad (6.1)$$

The same mechanism is adopted in this model with the exception that the liquid volume fraction in the slug is not obtained from the correlation proposed by Barnea, but from the Gregory et al.⁸ correlation given below:

$$E_{Ls} = \frac{1}{1 + \left(\frac{V_m}{8.66} \right)^{1.39}} \quad (6.2)$$

A transition from dispersed bubble flow to froth flow can also occur when the maximum volumetric packing density of the dispersed gas bubbles is exceeded (line D1 in Figure 6.1):

$$C_G = \frac{V_{SG}}{V_m} > 0.52 \quad (6.3)$$

If the criteria given by (6.1 and (6.3 are not satisfied, dispersed bubble flow is not possible and the possibility of stratified flow is examined next.

6.2.2 Stratified Flow

Determining the stability of the stratified flow regime requires the calculation of the liquid height, which can be obtained by writing the momentum balance equations for the gas and the liquid phases as was done by Taitel and Dukler³:

$$-A_L \left(\frac{dp}{dL} \right) - \tau_{wL} S_L + \tau_i S_i - \rho_L A_L \frac{g}{g_c} \sin \theta = 0 \quad (6.4)$$

$$-A_G \left(\frac{dp}{dL} \right) - \tau_{wG} S_G - \tau_i S_i - \rho_G A_G \frac{g}{g_c} \sin \theta = 0 \quad (6.5)$$

These can then be combined, eliminating the pressure gradient terms, and expressed in terms of the dimensionless liquid height, $\tilde{h}_L = h_L/D$, using the geometric relationships outlined by Taitel and Dukler. The shear stresses are given by the following relationships:

$$\tau_{wG} = \frac{f_G \rho_G V_G^2}{2g_c} \quad (6.6)$$

$$\tau_{wL} = \frac{f_L \rho_L V_L^2}{2g_c} \quad (6.7)$$

$$\tau_i = \frac{f_i \rho_G V_i |V_i|}{2g_c} \quad (6.8)$$

These definitions differ from those proposed by Taitel and Dukler mainly in that pipe roughness is not ignored when determining the friction factors. In addition, the definition for the gas/liquid interfacial shear does not require the assumption that the gas phase moves faster than the liquid phase (thereby assuming the shear stress to be based on the gas phase velocity alone). A new approach is also used when determining the liquid /wall interfacial friction factor, f_L , as discussed below.

The friction factor at the gas/wall interface Eq. (6.6) is determined from an approach similar to that used in single-phase flow with the actual pipe roughness and the following definition of Reynolds number:

$$\text{Re}_G = \frac{D_G \rho_G V_G}{\mu_G} \quad (6.9)$$

where D_G is the hydraulic diameter of the gas phase.

For the liquid/wall interface Eq. (6.7) it was found that the use of an approach similar to single phase flow is not appropriate and, instead, the following empirical relationship is used for calculating the wall/liquid interfacial friction factor:

$$f_L = 0.452 f_{SL}^{0.731} \quad (6.10)$$

The friction factor based on the superficial velocity, f_{SL} , is obtained from standard methods using the pipe roughness and the following definition of Reynolds number:

$$\text{Re}_{SL} = \frac{D \rho_L V_{SL}}{\mu_L} \quad (6.11)$$

During downhill flow, it is possible for the dense phase to flow faster than the lighter phase. For this reason, the definition of the gas/liquid interfacial shear Eq. (6.8) is based on the quantity $V_i = V_G - V_L$, which can become negative under certain conditions. The interfacial friction factor is calculated from the empirical relationship:

$$f_i = (0.004 + 0.5 \times 10^{-6} \text{Re}_{SL}) \text{Fr}_L^{1.335} \left\{ \frac{\rho_L D g}{\rho_G V_G^2} \right\} \quad (6.12)$$

The Froude number is defined as $\text{Fr}_L = \frac{V_L}{\sqrt{g h_L}}$.

Once the liquid height is known, the stability of the stratified flow pattern can be determined. The approach used by Taitel and Dukler, which uses an extension of the Kevin-Helmholtz wave stability theory, is also used in this model. This attempts to predict the gas velocity at which waves on the liquid surface are large enough to bridge the pipe:

$$V_G = \left(1 - \frac{h_L}{D} \right) \sqrt{\frac{(\rho_L - \rho_G) g A_G \cos \theta}{\rho_G \frac{dA_L}{dh_L}}} \quad (6.13)$$

This transition is represented by line S1 in Figure 6.1. The transition criterion expressed by (6.13) does not apply when $\theta = -90^\circ$. Experimental data, however, suggest that stratified-like annular flow can occur at such inclinations. In order to account for this, when $\cos \theta \leq 0.02$, $\cos \theta = 0.02$ is substituted in Eq. (6.13).

At steep downward inclinations, Barnea proposes a mechanism whereby stratified flow can change to annular, even at relatively low gas rates. This occurs when the liquid height is small and the liquid velocity is high. Liquid droplets are sheared off from the wavy interface and deposited on the upper pipe wall, eventually developing into an annular film. The condition for this type of transition to annular flow, shown as line S4 in Figure 6.1, is given as²:

$$V_L > \sqrt{\frac{g D (1 - \tilde{h}_L) \cos \theta}{f_L}} \quad (6.14)$$

It should be noted that f_L is calculated as per Eq. (6.10); not the definition proposed by Barnea.

At the higher upward pipe inclinations the predicted liquid height has the tendency, given the transition criterion of Eq. (6.13), to predict stratified flow where none is known to exist. For this reason, and to ensure continuity between flow pattern transitions, the present model limits stratified flow to horizontal and downhill angles only. This approach is also supported by the fact that stratified flow is only observed for small upward angles in large-diameter pipes.

Thus, when $\theta \leq 0$, if the gas phase velocity is less than the transitional value given by Eq. (6.13) and the liquid phase velocity is less than that of Eq. (6.14), the flow pattern is stratified. Although no distinction is made in this model between stratified smooth and stratified wavy flow for the purposes of determining pressure drop and liquid volume fraction, the transition between these

two regimes is considered in flow pattern predictions. Taitel and Dukler propose that waves will form on the liquid surface once the gas velocity is increased beyond (line S2 in Figure 6.1):

$$V_G \geq \sqrt{\frac{4\mu_L(\rho_L - \rho_G)g \cos \theta}{s\rho_L\rho_G V_L}} \quad (6.15)$$

The sheltering coefficient, s , is given as 0.01. In Xiao et al.⁵ and in the present model, s is taken as 0.06, based on a study by Andritsos⁹. This value is said to be more suitable, especially for gas flow with high viscosity liquids.

During downflow, waves can develop on the flowing liquid independent of interfacial shear from the gas flow. The criterion for the appearance of waves can be expressed in terms of a critical Froude number which varies from 0.5 to 2.2 depending on roughness and whether the flow is laminar or turbulent. Barnea² recommends a limiting value of 1.5 for the critical Froude number. When interfacial effects are considered in the calculation of the liquid height, this limit can predict smooth flow even at high liquid rates where the flow is known to be wavy. Reducing the limit to 1.4 appears to resolve this problem. Thus the transition from stratified smooth to wavy flow based on this mechanism is (line S3 in Figure 6.1):

$$\text{Fr} = \frac{V_L}{\sqrt{gh_L}} > 1.4 \quad (6.16)$$

6.2.3 Annular-Mist Flow

The treatment of the annular-mist flow regime is similar to the approach used for stratified flow and is based on the work of Taitel and Dukler³ and Oliemans et al.¹⁰. The model is based on the assumption of a constant film thickness and accounts for the entrainment of the liquid in the gas core. Slip between the liquid droplets in the gas core and the gas phase is not accounted for. Momentum balance on the liquid film and gas core with liquid droplets yields:

$$-A_f \left(\frac{dp}{dL} \right) - \tau_{wL} S_L + \tau_i S_i - \rho_L A_f \frac{g}{g_c} \sin \theta = 0 \quad (6.17)$$

$$-A_c \left(\frac{dp}{dL} \right) - \tau_i S_i - \rho_c A_c \frac{g}{g_c} \sin \theta = 0 \quad (6.18)$$

The geometric parameters can be expressed in terms of the dimensionless liquid film thickness, $\tilde{\delta}_L = \delta_L/D$, and the liquid fraction entrained, FE . The shear stresses are given by:

$$\tau_{wL} = \frac{f_f \rho_L V_f^2}{2g_c} \quad (6.19)$$

$$\tau_i = \frac{f_i \rho_c (V_c - V_f) |V_c - V_f|}{2g_c} \quad (6.20)$$

The friction factor for the liquid film is computed using any of the standard correlations with the pipe roughness and the film Reynolds number as expressed by:

$$\text{Re}_f = \frac{D_f \rho_L V_f}{\mu_L} \quad (6.21)$$

In order to solve Eqs. (6.17) and (6.18), two additional quantities need to be determined: the interfacial friction factor, f_i , and the liquid fraction entrained, FE . These are determined empirically and are given by:

$$\frac{FE}{1-FE} = 0.735 N_B^{0.074} \left(\frac{V_{SG}}{V_{SL}} \right)^{0.2} \quad (6.22)$$

$$\frac{f_i}{f_c} = 0.24 \left\{ \frac{\sigma}{\rho_c V_c^2 D_c} \right\}^{0.085} \text{Re}_f^{0.305} \quad (6.23)$$

Where the dimensionless number, N_B , is defined as

$$N_B = \frac{\mu_L^2 V_{SG}^2 \rho_G}{\sigma^2 \rho_L} \quad (6.24)$$

Having determined the liquid film thickness, it is now possible to test for the presence of annular-mist flow. Barnea² presents a model for the transition from annular flow based on two conditions. The same mechanisms are used in the present model, although they are revised to account for the differences in the modeling assumptions.

The first of the transitions proposed by Barnea is based on the observation that the minimum interfacial shear stress is associated with a change in the direction of the velocity profile in the film. When the velocity profile becomes negative stable annular flow cannot be maintained and the transition to intermittent flow occurs. This transition mechanism is only relevant during uphill flow. The minimum shear stress condition may be determined by setting $\partial \tilde{\delta}_L / \partial \tilde{\delta}_L = 0$.

$$2f_f \frac{\rho_L}{\rho_L - \rho_c} \frac{V_{SL}^2 (1-FE)^2}{gD \sin \theta} = \frac{E_f^3 (1 - \frac{3}{2} E_f)}{2 - \frac{3}{2} E_f} \quad (6.25)$$

The liquid fraction in the film is given by:

$$E_f = \frac{A_f}{A} = 4\tilde{\delta}_L (1 - \tilde{\delta}_L) \quad (6.26)$$

Eq. (6.25) can be solved using an iterative procedure to obtain the liquid film height at which the minimum shear stress occurs, $\tilde{\delta}_{L_{min}}$ (line A1 in Figure 6.1).

The second mechanism proposed by Barnea for annular flow instability occurs when the supply of liquid in the film is sufficient to cause blockage of the gas core by bridging the pipe. This is said to take place when the in situ volume fraction of liquid exceeds one half of the value associated with the maximum volumetric packing density of uniformly sized gas bubbles (0.52). Hence, the transition from annular flow occurs when (line A2 in Figure 6.1):

$$E_L \geq \frac{1}{2} (1 - 0.52) \text{ or } E_L \geq 0.24 \quad (6.27)$$

6.2.4 Bubble Flow

When the liquid fraction in the slug Eq. (6.2) is greater than 0.48 and the stratified, annular and dispersed bubble flow regimes have been eliminated, the flow will either be intermittent, froth or bubble flow.

Bubble flow is encountered in steeply inclined pipes and is characterized by a continuous liquid phase containing a dispersed phase of mostly spherical gas bubbles. It can exist if both of the following conditions are satisfied:

1. The Taylor bubble velocity exceeds the bubble velocity. This is satisfied in large diameter pipes (Taitel et al.¹¹) when

$$D > 19 \left[\frac{(\rho_L - \rho_G)\sigma}{\rho_L^2 g} \right]^{\frac{1}{2}} \quad (6.28)$$

2. The angle of inclination is large enough to prevent migration of bubbles to the top wall of the pipe (Barnea et al.¹²):

$$\cos \theta \leq \frac{3}{4\sqrt{2}} V_b^2 \left(\frac{C_l \gamma^2}{g d_b} \right) \quad (6.29)$$

The lift coefficient, C_l , ranges from 0.4 to 1.2, the bubble distortion (from spherical) coefficient, γ , ranges from 1.1 to 1.5 and a bubble size, d_b , between 4 and 10mm is recommended. For this model, C_l is taken as 0.8, γ as 1.3 and a bubble diameter of 7 mm is used. The bubble swarm rise velocity in a stagnant liquid, V_b , is given by¹³:

$$V_b = 1.41 \left[\frac{g(\rho_L - \rho_G)\sigma}{\rho_L^2} \right]^{\frac{1}{4}} \sin \theta \quad (6.30)$$

When both of the above conditions are satisfied, bubble flow is observed even at low liquid rates where turbulence does not cause bubble breakup. The transition to bubble flow from intermittent flow as suggested by Taitel et al.¹¹ occurs when the gas void fraction (during slug flow) drops below the critical value of 0.25 (line I3 in Figure 6.1). The calculation of the gas void fraction for slug flow is discussed below.

6.2.5 Intermittent Flow

The intermittent flow model used here includes the slug and elongated bubble flow patterns. It is characterized by alternating slugs of liquid trailed by long bubbles of gas. The liquid slug may contain dispersed bubbles and the gas bubbles have a liquid film below them.

As stated above, a transition from intermittent flow occurs when the liquid fraction in the slug exceeds the value associated with the maximum volumetric packing density of the dispersed bubbles ((6.1, line I1 in Figure 6.1). The same mechanism can occur at low liquid rates when sufficient liquid is not available for slug formation. To account for this situation, an additional transition criterion is imposed (line I4 in Figure 6.1).

$$E_L \leq 0.24 \quad (6.31)$$

The liquid volume fraction calculated for slug flow is discussed below. Although it is not treated as a separate flow pattern for the purposes of phase volume fractions and pressure drop determination, the elongated bubble flow regime is defined here as the portion of intermittent flow for which the liquid slug contains no dispersed bubbles of gas. This condition is arbitrarily represented in the model by the region where $E_{L_s} \geq 0.90$ (line I2 in Figure 6.1).

The liquid volume fraction may be determined by writing an overall liquid mass balance over a slug-bubble unit. Assuming that the flow is incompressible and a uniform depth for the liquid film¹⁴:

$$E_L = \frac{E_{L_s}V_t + V_{Gdb}(1 - E_{L_s}) - V_{SG}}{V_t} \quad (6.32)$$

V_{Gdb} represents the velocity of the dispersed bubbles, V_t is the translational velocity of the slug, and E_{L_s} is the volume fraction liquid in the slug body Eq. (6.2). All of these quantities need to be determined from empirical correlations.

The translational velocity of the elongated bubbles is given by Bendiksen¹⁵ as:

$$V_t = C_0 V_m + V_d \quad (6.33)$$

The parameter C_0 is a distribution coefficient related to the velocity and concentration profiles in dispersed systems and under special conditions is related to the inverse of the Bankoff K factor. Zuber and Findlay¹⁶ have confirmed empirically its application to other flow patterns, including slug and annular flow. Nicklin et al.¹⁷, in their study of the rise velocity of Taylor bubbles, have found that for liquid Reynolds numbers greater than 8,000, $C_0 = 1.2$, whereas at lower Reynolds numbers C_0 approached 2.0. It is generally taken to be 1.2, although for this analysis it is determined from the following empirically derived correlation:

$$C_0 = (1.64 + 0.12 \sin \theta) \text{Re}_{mL}^{-0.031} \quad (6.34)$$

The modified Reynolds number in Eq. (6.34) is based on the mixture velocity and liquid properties:

$$\text{Re}_{mL} = \frac{\rho_L V_m D}{\mu_L} \quad (6.35)$$

The elongated bubble drift velocity, V_d , can be calculated from the Zuboski¹⁸ correlation:

$$V_d = f_m V_{d_\infty} \quad (6.36)$$

Where $f_m = 0.316 \sqrt{\text{Re}_\infty}$ for $f_m < 1$, otherwise $f_m = 1$, and

$$\text{Re}_\infty = \frac{\rho_L V_{d_\infty} D}{2\mu_L} \quad (6.37)$$

Bendiksen¹⁵ gives the elongated bubble drift velocity at high Reynolds numbers as:

$$V_{d_\infty} = V_{dh_\infty} \cos \theta + V_{dv_\infty} \sin \theta \quad (6.38)$$

The drift velocity of elongated bubbles in a horizontal system at high Reynolds numbers is given by Weber¹⁹ as:

$$V_{dh_{\infty}} = \left[0.54 - \frac{1.76}{Bo^{0.56}} \right] \sqrt{\frac{gD(\rho_L - \rho_G)}{\rho_L}} \quad (6.39)$$

The Bond number, $Bo = \frac{(\rho_L - \rho_G)}{\sigma} gD^2$

The drift velocity of elongated bubbles in a vertical system at high Reynolds numbers is obtained from a modified form of the Wallis²⁰ correlation

$$V_{dv_{\infty}} = 0.345(1 - e^{-\beta}) \sqrt{\frac{gD(\rho_L - \rho_G)}{\rho_L}} \quad (6.40)$$

The coefficient, β , is given by:

$$\beta = Bo e^{(3.278 - 1.424 \ln Bo)} \quad (6.41)$$

Finally, the volume fraction liquid Eq. (6.32) can be calculated once the velocity of the dispersed bubbles in the liquid slug is obtained from:

$$V_{Gdb} = C_0 V_m + V_b \quad (6.42)$$

C_0 in Eq. (6.42) is determined from Eq. (6.34) and the rise velocity of the dispersed bubbles is calculated from²¹:

$$V_b = 1.53 \left[\frac{g(\rho_L - \rho_G)\sigma}{\rho_L^2} \right]^{\frac{1}{4}} \sin \theta \quad (6.43)$$

The empirical nature of the correlations used for determining the liquid volume fraction Eq. (6.32) requires that certain limits be imposed on the calculated values. The first such condition affects Eq. (6.42) where it is possible, under certain downflow conditions for the calculated value of V_{Gdb} to become negative. In these situations, V_{Gdb} is set to zero. In other situations, it is possible for Eq. (6.32) to yield values for E_L that are greater than 1.0. In these cases, E_L is set equal to C_L .

When none of the transition criteria listed above are met, the flow pattern is designated as "Froth" implying a transitional state between the other flow regimes.

6.3 Calculation of Pressure Drop and Liquid Volume Fraction

Once the flow pattern has been determined, the calculation of the pressure drop and phase volume fractions can be determined as detailed below.

6.3.1 Dispersed Bubble Flow

The calculation of the liquid volume fraction in dispersed bubble flow follows the procedure used for the dispersed bubbles in the slug in intermittent flow. Thus,

$$V_{Gdb} = C_0 V_m + V_b \quad (6.44)$$

C_0 is determined from the empirical correlation given in Eq. (6.34), and the rise velocity of the dispersed bubbles, V_b , is calculated from Eq. (6.43). The volume fraction is then obtained from:

$$E_L = 1 - \frac{V_{SG}}{V_{Gdb}} \quad (6.45)$$

If $V_{Gdb} \leq 0$, the volume fraction is then obtained from:

$$E_L = 1 - \frac{V_{SG}}{C_0 V_m} \quad (6.46)$$

In cases where the value of E_L calculated by Eq. (6.45) or Eq. (6.46) is greater than 1.0, E_L is set equal to C_L .

Once the liquid volume fraction is known, the pressure gradient is determined from:

$$-\left(\frac{dp}{dL}\right) = \frac{2f_m V_m^2 \rho_m}{g_c D} + \rho_m \frac{g}{g_c} \sin \theta \quad (6.47)$$

The friction factor, f_m , is obtained from standard methods using the pipe roughness and the following Reynolds number:

$$\text{Re}_m = \frac{D \rho_m V_m}{\mu_m} \quad (6.48)$$

The mixture density and viscosity are calculated in the usual way:

$$\rho_m = E_L \rho_L + E_G \rho_G \quad (6.49)$$

$$\mu_m = E_L \mu_L + E_G \mu_G \quad (6.50)$$

6.3.2 Stratified Flow

The liquid volume fraction during stratified flow is simply, from geometric considerations:

$$E_L = \frac{A_L}{A} \quad (6.51)$$

The pressure gradient is obtained from either Eq. (6.4) or Eq. (6.5).

6.3.3 Annular-Mist Flow

As is the case for stratified flow, the volume fraction liquid during annular-mist flow is determined from geometric considerations once the liquid film thickness is known:

$$E_L = 1 - (1 - 2\tilde{\delta}_L)^2 \frac{V_{SG}}{V_{SG} + FE V_{SL}} \quad (6.52)$$

The pressure gradient is obtained from either Eq. (6.17) or Eq. (6.18).

6.3.4 Bubble Flow

The volumetric gas fraction during bubble flow is obtained from:

$$E_G = \frac{V_{SG}}{V_t} \quad (6.53)$$

The translational bubble velocity is defined as:

$$V_t = C_0 V_m + V_b \quad (6.54)$$

Zuber and Findlay¹⁶ have shown that the distribution parameter, C_0 , for dispersed systems can range from 1.0 to 1.5, the higher values being associated with high bubble concentrations and high velocities at the center line (laminar flow). When the flow is turbulent and the velocity and concentration profiles are flat C_0 approaches 1.0. For the present method, C_0 is taken as 1.2. The bubble swarm rise velocity in a stagnant liquid, V_b , is given by (6.30). The value of E_G thus obtained, is limited to the range:

$$0 \leq E_G \leq C_G = \frac{V_{SG}}{V_m} \quad (6.55)$$

The pressure gradient is given by:

$$-\left(\frac{dp}{dL}\right) = \frac{2f_{mL}V_m^2\rho_m}{g_c D} + \rho_m \frac{g}{g_c} \sin \theta \quad (6.56)$$

The friction factor, f_{mL} , is obtained from standard methods using the pipe roughness and the following Reynolds number:

$$\text{Re}_{mL} = \frac{D\rho_L V_m}{\mu_L} \quad (6.57)$$

6.3.5 Intermittent Flow

The calculation of the volume fraction liquid for intermittent flow has already been described Eq. (6.32). The pressure drop may be obtained by writing the momentum balance over a slug-bubble unit:

$$-\left(\frac{dp}{dL}\right) = \rho_m \frac{g}{g_c} \sin \theta + \frac{1}{L_u} \left[L_s \left(\frac{\tau_{Ls} \pi D}{A} \right) + L_f \left(\frac{\tau_{Lf} S_{Lf} + \tau_{Gdb} S_{Gdb}}{A} \right) \right] \quad (6.58)$$

Unfortunately, no reliable methods exist for the calculation of the slug length, L_s , nor for the length of the bubble region, L_f . Furthermore, although it is known that the frictional pressure gradient in the gas bubble is normally small compared to that in the liquid slug, no reliable method is available for calculating it. Xiao et al.⁵ have modeled the bubble region by assuming it to be analogous to stratified flow. This treatment contradicts observations made in the laboratory. In view of these uncertainties, the following simple approach is selected:

$$-\left(\frac{dp}{dL}\right) = \rho_m \frac{g}{g_c} \sin \theta + \eta \left(\frac{dp}{dL} \right)_{f_{SL}} + (1 - \eta) \left(\frac{dp}{dL} \right)_{f_{AM}} \quad (6.59)$$

Here, the quantity η is an empirically determined weighting factor related to the ratio of the slug length to the total slug unit length, $\frac{L_s}{L_u}$ and is calculated from:

$$\eta = C_L^{(0.75-E_L)} \quad (6.60)$$

with the condition that $\eta \leq 1.0$.

The frictional pressure gradient for the slug portion is obtained from:

$$\left(\frac{dp}{dL} \right)_{fr_{sl}} = 2 \frac{f_{mL} V_m^2 \rho_m}{gD} \quad (6.61)$$

The friction factor, f_{mL} , is calculated from standard methods using the pipe roughness and the Reynolds number given by Eq. (6.57).

The term that still needs to be defined in Eq. (6.59) is the frictional pressure gradient calculated for annular-mist flow. This is obtained by using the liquid fraction given by Eq. (6.32) to determine the liquid film height, assuming the flow pattern to be annular-mist. Thus, from Eq. (6.52),

$$\tilde{\delta}_L = \frac{1}{2} \left[1 - \sqrt{(1 - E_L) \frac{(FE V_{SL} + V_{SG})}{V_{SG}}} \right] \quad (6.62)$$

The frictional pressure gradient based on annular-mist flow is then calculated from:

$$\left(\frac{dp}{dL} \right)_{fr_{AM}} = \frac{4\tau_{wL}}{D} \quad (6.63)$$

The shear stress τ_{wL} is obtained from Eq. (6.19). Note that Eq. (6.63) is obtained by adding Eqs. (6.17) to (6.18) (to eliminate the interfacial component) and removing the hydrostatic term.

When the calculated film height Eq. (6.62) is less than 1×10^{-4} , a simple homogeneous model with slip is used, where:

$$\left(\frac{dp}{dL} \right)_{fr_{AM}} = \frac{2f_m V_m^2 \rho_m}{g_c D} \quad (6.64)$$

The friction factor, f_m , is obtained from standard methods using the pipe roughness and Reynolds number given in Eq. (6.48). The mixture density and viscosity are obtained from Eqs. (6.49) and (6.50), respectively.

6.3.6 Froth Flow

Froth flow represents a transition zone between dispersed bubble flow and annular-mist flow and between slug flow and annular-mist. The approach used in this model is to interpolate between the appropriate boundary regimes in order to determine the transition values of the in situ liquid volume fraction and pressure drop. This involves a number of iterative procedures in order to determine the superficial gas velocities at the dispersed bubble, annular-mist and slug transitions to froth. Once V_{SG} at each transition is known, the volume fraction and pressure drop values *at the transitions* are calculated and a log-log interpolation between these values is made for each quantity.

6.4 Results

The model's overall performance has been evaluated using the following approaches.

- a) The behavior of the model was examined over a wide range of flow rates and fluid properties using three-dimensional surface plots. This was done over the complete range of upward and downward pipe inclinations and both, pressure gradient and volumetric liquid fraction were analyzed.
- b) Data were extracted from the Stanford Multiphase Flow Database for which pressure gradient, holdup and flow pattern observations were available. This resulted in a total of 5,951 measurements consisting of variations in fluid properties, pipe diameters, and upward as well as downward inclinations. The model was then compared with these experimental observations.
- c) Finally, these same experimental data were analyzed using a number of existing methods and the results compared to the new model.

Flow pattern maps and three dimensional surface plots are shown in Figures 6.4 to 6.27 for an air/water system at standard conditions and for an oil/gas system at reservoir conditions (see Table 6.1). The pipe inclinations shown include horizontal, 10° upward, vertical upflow ($+90^\circ$), and 10° downward. Each plot covers a range of superficial gas velocity of 0.01 ft/sec to 500 ft/sec and of superficial liquid velocity from 0.01 ft/sec to 100 ft/sec.

The coloring of the three dimensional plots is consistent with that of the flow pattern maps so as to show the location of the flow pattern transitions. The superficial velocity axes appear on the X-Y plane with the appropriate parameter (pressure gradient or liquid volume fraction) plotted on the vertical axis. Over one hundred different gas and liquid rates have been calculated per plot, equating to over 10,000 calculated points. This was done to insure that the model behaves predictably over the entire practical range of flow rates and pipe inclinations. The effect of fluid properties on flow pattern transitions can easily be seen in these plots. Although some discontinuities in pressure gradient and liquid volume fraction are present at the transitions from stratified flow, overall, the model exhibits generally smooth behavior and consistent trends between flow patterns.

The distribution of experimental data points according to angle of inclination is shown in Figure 6.28. The convention used in expressing the range of inclinations in this figure is that the lower number in the specified range is inclusive, whereas the higher number is not. Thus, " 0° to 10° " implies the range where $0^\circ \leq \theta < 10^\circ$. Although more than half of the data fall in the range of 0° to 10° inclination, it can be seen that there is also a fair sampling (1,164 points, or $\sim 20\%$) of downhill data.

The model predictions for liquid volume fraction are plotted against the experimental measurements in Figure 6.29. Figure 6.30 shows a similar plot for the pressure gradient calculations. The model is able to predict the in situ liquid volume fraction to within an accuracy of 15% in 3,663 of the 5,951 cases (62%). This is shown in Figure 6.31, where these numbers are compared with

other methods. The pressure gradient is predicted to the same accuracy for 2,567 cases (43%), as shown in Figure 6.31.

Some of the methods with which comparisons are being made are limited to specific ranges of pipe inclination. To make the comparisons of Figure 6.31 more meaningful, they are shown grouped according to downward, near-horizontal and upward pipe inclination in Figure 6.32, for the volume fraction liquid, and in Figure 6.33, for the pressure gradient.

6.5 Comments

The empirical correlations introduced in this paper, namely:

- Eqs. (6.10) and (6.12) for stratified flow,
- Eqs. (6.22) and (6.23) for annular-mist flow,
- Eqs. (6.34), (6.40) and (6.60) for intermittent flow,

were developed in accordance with the following two objectives.

- a) Accurately reflect the expected/observed behavior of the quantity being estimated.
- b) Ensure that the correlation's behavior results in smooth transitions between adjacent flow patterns.

The application of both of these criteria involved relying on statistical analysis of differences between predicted and experimental values as well as the study of numerous surface plots (such as those shown in Figures 6.4 to 6.27). This did not always result in the "statistically best" correlation being adopted.

Solving the momentum balance equations in stratified flow (Eqs. (6.4) and (6.5)) and annular-mist flow (Eqs. (6.17) and (6.18)) poses certain problems because of the presence of multiple roots. Primarily, it is necessary to determine which of the roots is the physical one. Xiao et al.⁵ assume that it is the lowest root. There does not however seem to exist a clear rationale for this assumption. Figure 6.2 shows the variation of liquid height versus superficial gas velocity at a fixed superficial liquid rate for an air/water system at standard conditions (2.047" I.D. pipe, at 2° upward inclination). It can be seen that in the range where multiple roots occur ($V_{SG} = 57.144$ to 112.67 ft/sec), the selection of the lowest root versus the highest root results in significant changes to the predicted liquid height. This large variation in liquid height based on different roots suggests that the selection of one root over the other simply affects the value of V_{SG} at which a transition to another flow pattern occurs. In order to prevent discontinuities, it is important to ensure that, whether the lowest or the highest root is used, the same root is used in all the calculations. In the present model, the lowest root is selected for the ensuing calculations. Having to determine all of the roots presents a further complication as can be seen by examining Figure 6.2 when $V_{SG} = 58.0$ ft/sec. It is seen that three roots exist under these conditions: 0.0586, 0.0646, and 0.508. The lower two roots can only be detected by investigating values of $\tilde{h}_L$ within less than 0.005 of each other. For the range of validity of the solutions (0 to 1.0) this could require over 200 iterations. For certain combinations of fluid rates and pipe inclination the required resolution of liquid height or liquid film thickness is of the order of 0.001, which can require over 10,000 iterations.

A final note is warranted regarding discontinuities that arise from friction factor calculations when the flow changes from laminar to turbulent. The traditional approach is to use the laminar flow friction factor when the Reynolds number is less than 2,000. In the present model, the approach followed is to use the turbulent friction factor wherever it is greater than the laminar flow value. This results in a smoother correlation and since the application of single-phase friction factor correlations to multiphase flow situations is, at best, arbitrary, the approach is believed to be reasonable.

6.6 Conclusions

A new mechanistic model has been presented which is applicable to all conditions commonly encountered in the petroleum industry. The model incorporates roughness effects as well as liquid entrainment, both of which are not accounted for by previous models. The model has undergone extensive testing and has proven to be more robust than existing models and is applicable over a more extensive range of conditions.

The empirical correlations that are necessary within the model can only be improved with accurate and consistent data over a wide range of conditions of commercial interest. To this end, efforts are in progress to obtain additional data in order to expand the Stanford Multiphase Flow Database.

Further testing of the new model will be undertaken which will include as much actual field data as can be obtained. It is expected that results from these tests will more adequately demonstrate the ability of the model to predict reasonably accurate pressure drops and holdup under operating conditions.

Acknowledgements

The work described in this paper has been made possible through the support of the Reservoir Simulation Industrial Affiliates Program at Stanford University (SUPRI-B) and the Stanford Project on the Productivity and Injectivity of Horizontal Wells (SUPRI-HW). Portions of this research have also been supported by the U.S. Department of Energy contract DE-FG22-93BC14862. The development of the model has also greatly benefited through numerous discussions with Mr. Liang-Biao Ouyang of Stanford University.

Nomenclature

A	Cross-sectional area
C_0	Velocity distribution coefficient
D	Pipe internal diameter
E	In situ volume fraction
FE	Liquid fraction entrained
G	Acceleration due to gravity
h_L	Height of liquid (Stratified flow)
L	Length
P	Pressure
Re	Reynolds number
S	Contact perimeter
V_{SG}	Superficial gas velocity
V_{SL}	Superficial liquid velocity
δ_L	Liquid film thickness (Annular-Mist)
ε	Pipe roughness
η	Pressure gradient weighting factor (intermittent flow)
θ	Angle of inclination
μ	Viscosity
ρ	Density
σ	Interfacial (surface) tension
τ	Shear stress
$\tilde{x}$	Dimensionless quantity, x

Subscripts

b	relating to the gas bubble
c	relating to the gas core
f	relating to the liquid film
db	relating to the dispersed bubbles
G	relating to the gas phase
L	relating to the liquid phase
m	relating to the mixture
SG	based on superficial gas velocity
s	relating to the liquid slug
SL	based on superficial liquid velocity
wL	relating to the wall-liquid interface
wG	relating to the wall-gas interface

References

1. Aziz, K. and Petalas, N., "New PC-Based Software for Multiphase Flow Calculations," SPE 28249, SPE Petroleum Computer Conference, Dallas, 31 July-3 August, 1994.
2. Barnea, D. "A Unified Model for Predicting Flow-Pattern transitions for the Whole Range of Pipe Inclinations," Int. J. Multiphase Flow, 13, No. 1, 1-12 (1987).
3. Taitel, Y., and Dukler, A. E. "A Model for predicting Flow Regime Transitions in Horizontal and Near Horizontal Gas-Liquid Flow," AIChE Journal, 22, 47 (1976).
4. Ansari, A. M., Sylvester, A. D., Sarica, C., Shoham, O., and Brill, J. P., "A Comprehensive Mechanistic Model for Upward Two-Phase Flow in Wellbores," SPE Prod. & Facilities, pp. 143-152, May 1994.
5. Xiao, J. J., Shoham, O., Brill, J. P., "A Comprehensive Mechanistic Model for Two-Phase Flow in Pipelines," paper SPE 20631, 65th ATC&E of SPE, New Orleans, September 23-26, 1990.
6. Petalas, N., and Aziz, K., "Development and Testing of a New Mechanistic Model for Multiphase Flow in Pipes," Proceedings of the ASME Fluids Division Summer Meeting, Volume 1, FED-Vol. 236, pp. 153-159, July, 1996.
7. Petalas, N., and Aziz, K., "Stanford Multiphase Flow Database - User's Manual," Version 0.2, Petroleum Engineering Dept., Stanford University, 1995.
8. Gregory, G. A., Nicholson, M.K. and Aziz, K., "Correlation of the Liquid Volume Fraction in the Slug for Horizontal Gas-Liquid Slug Flow," Int. J. Multiphase Flow, 4, 1, pp. 33-39 (1978).
9. Andritsos, N., "Effect of Pipe Diameter and Liquid Viscosity on Horizontal Stratified Flow," Ph.D. Dissertation, U. of Illinois at Champaign-Urbana (1986).
10. Oliemans, R. V. A., Pots, B. F., and Trope, N., "Modeling of Annular Dispersed Two-Phase Flow in Vertical Pipes," Int. J. Multiphase Flow, 12, No. 5, 711-732 (1986).
11. Taitel, Y., Barnea, D. and Dukler, A. E. "Modeling Flow Pattern Transitions for Steady Upward Gas-Liquid Flow in Vertical Tubes," AIChE Journal, 26, pp. 345-354 (1980).
12. Barnea, D., Shoham, O., Taitel, Y. and Dukler, A.E., "Gas-Liquid Flow in Inclined Tubes: Flow Pattern Transitions for Upward Flow," Chem. Eng. Sci., 40, 1 pp. 131-136 (1985).
13. Zuber, N., Staub, F.W., Bijwaard, G., and Kroeger, P.G., "Steady State and Transient Void Fraction in Two-Phase Flow Systems," 1, Report EURAEC-GEAP-5417, General Electric Co., San Jose, California, January 1967.
14. Govier, G. W. and Aziz, K. "The Flow of Complex Mixtures in Pipes," Van Nostrand, Reinhold (1972), reprinted by Robert E. Kriger Publishing Co., Huntington, New York, 1977.
15. Bendiksen, K. H. "An Experimental Investigation of the Motion of Long Bubbles in Inclined Pipes," Int. J. Multiphase Flow, 10, pp 1-12 (1984).
16. Zuber, N., and Findlay, J.A., "Average Volumetric Concentration in Two-Phase Flow Systems," J. Heat. Transfer, Trans. ASME, Ser. C, 87, pp. 453-468 (1965).

17. Nicklin, D. J., Wilkes, J. O., and Davidson, J. F., "Two Phase Flow in Vertical Tubes," Trans. Inst. Chem. Engrs., 40, pp. 61-68, (1962).
18. Zukoski, E.E., "Influence of Viscosity, Surface Tension, and Inclination Angle on Motion of Long Bubbles in Closed Tubes," J. Fluid Mech., 25, pp. 821-837 (1966).
19. Weber, M. E., "Drift in Intermittent Two-Phase Flow in Horizontal Pipes," Canadian J. Chem. Engg., 59, pp. 398-399, June 1981.
20. Wallis, G. B. "One-Dimensional Two-Phase Flow," McGraw-Hill, New York, 1969.
21. Harmathy, T.Z., "Velocity of Large Drops and Bubbles in Media of Infinite or Restricted Extent," AIChE J., 6, pg. 281 (1960).

Table 6.1. System Properties for Flow Pattern Maps

	Air/Water System	Oil/Gas System
Pipe diameter	2.047 in	6.18 in.
Gas Density	.08 lb/ft ³	8.139 lb/ft ³
Liquid Density	62.4 lb/ft ³	52.53 lb/ft ³
Gas Viscosity	0.01 cP	0.018 cP
Liquid Viscosity	1.0 cP	2.757 cP
Interfacial Tension	72.4 dyne/cm	20 dyne/cm
(Absolute) Pipe Roughness	0.00015 ft	0.01 ft [†]

† This high value of roughness is used to represent an open-hole well completion.

Figure 6.1. Transitions used in flow pattern determination

Figure 6.2. Stratified Flow: Multiple Roots in Liquid Height Calculation for Increasing V_{SG} (Air/Water System)

Figure 6.3. Flow pattern determination for mechanistic model

Figure 6.4. Flow pattern map for air/water system at 0° inclination (horizontal)

Figure 6.7. Flow pattern map for oil/gas system at 0° inclination (horizontal)

Figure 6.5. Frictional pressure gradient for air/water system at 0° inclination (horizontal)

Figure 6.8. Frictional pressure gradient for oil/gas system at 0° inclination (horizontal)

Figure 6.6. Liquid volume fraction for air/water system at 0° inclination (horizontal)

Figure 6.9. Liquid volume fraction for oil/gas system at 0° inclination (horizontal)

Figure 6.10. Flow pattern map for air/water system at 10° upward inclination

Figure 6.13. Flow pattern map for oil/gas system at 10° upward inclination

Figure 6.11. Frictional pressure gradient for air/water system at 10° upward inclination

Figure 6.14. Frictional pressure gradient for oil/gas system at 10° upward inclination

Figure 6.12. Liquid volume fraction for air/water system at 10° upward inclination

Figure 6.15. Liquid volume fraction for oil/gas system at 10° upward inclination

Figure 6.16. Flow pattern map for air/water system at 90° upward inclination

Figure 6.17. Frictional pressure gradient for air/water system at 90° upward inclination

Figure 6.18. Liquid volume fraction for air/water system at 90° upward inclination

Figure 6.19. Flow pattern map for oil/gas system at 90° upward inclination

Figure 6.20. Frictional pressure gradient for oil/gas system at 90° upward inclination

Figure 6.21. Liquid volume fraction for oil/gas system at 90° upward inclination

Figure 6.22. Flow pattern map for air/water system at 10° downward inclination

Figure 6.25. Flow pattern map for oil/gas system at 10° downward inclination

Figure 6.23. Frictional pressure gradient for air/water system at 10° downward inclination

Figure 6.26. Frictional pressure gradient for oil/gas system at 10° downward inclination

Figure 6.24. Liquid volume fraction for air/water system at 10° downward inclination

Figure 6.27. Liquid volume fraction for oil/gas system at 10° downward inclination

Figure 6.28. Distribution of inclination angle for experimental data

Figure 6.31. Comparison of selected methods' ability to predict experimental volume fraction liquid and pressure gradient to within 15% accuracy

Figure 6.29. Mechanistic model volume fraction liquid calculations compared with experimental data

Figure 6.32. Comparison of selected methods' ability to predict experimental volume fraction liquid to within 15% accuracy (grouped by angle of inclination)

Figure 6.30. Mechanistic model pressure gradient calculations compared with experimental data

Figure 6.33. Comparison of selected methods' ability to predict experimental pressure gradient to within 15% accuracy (grouped by angle of inclination)

7. A Comprehensive Reservoir/Wellbore Model for Horizontal Wells

by V.R. Penmatcha and K. Aziz*

Abstract

Comprehensive, three-dimensional, anisotropic and transient, reservoir/wellbore coupling models are developed for infinite-conductivity and finite-conductivity wellbores to calculate the productivity of a horizontal well placed in a box-shaped drainage volume. The finite-conductivity well model considers frictional and accelerational pressure drops in the wellbore. It also takes into account the effects of fluid inflow into the well. These models are semianalytical in nature and can be used under transient or pseudosteady state flow conditions. The infinite-conductivity well model is compared with the uniform flux model to show the differences in well productivity calculations. The effects of wellbore pressure drop on well productivity are shown using the finite-conductivity well model. These models can provide reference solutions to calculate well indices or to check the accuracy of a numerical simulator. Because of their analytical nature they are easier and faster to use than numerical simulators. Using super-position in space and time, these models can be run under a variety of constraints. We believe that our finite-conductivity well model is more general than any published previously.

7.1 Introduction

Fictional effects can be important in long horizontal wells in high permeability reservoirs. In such cases the drawdowns are low and they are of the same order of magnitude as the frictional pressure drop in the well. If wellbore pressure drop is not taken into account in such cases, the productivity of a well can be grossly overestimated. In reservoirs with an aquifer or a gas cap, low drawdowns need to be maintained in order to prevent early breakthrough of water or gas. But these low drawdowns can cause the wellbore pressure drop to be very important. Ignoring wellbore pressure drop in such a scenario would make the engineer assume that the encroachment of water/gas is uniform towards the horizontal well. But in reality it is very skewed and fluids tend to breakthrough first at the heel of the well. The only way to capture these effects is by including wellbore pressure drop in the calculations.

Reservoir/wellbore coupling models available in the literature like those of Dikken¹, Landman², Novy³, and Penmatcha *et al.*⁴ represent the flow in the reservoir using a productivity index. This makes these models one-dimensional in nature and they cannot capture all the three-dimensional flow effects that can take place in the reservoir. Works of Ozkan *et al.*⁵ and Sarica *et*

* This paper (SPE 39521) was presented at the 1998 SPE India Oil and Gas Conference and Exhibition held in New Delhi, India, 7-9 April 1998.

al.⁶, though they describe flow in the reservoir using three-dimensional models, assume that the reservoir is infinite-acting and hence these models cannot be used to evaluate the effects of the wellbore pressure drop under pseudosteady state conditions. All these past works have considered only frictional pressure drop in the wellbore and they have ignored accelerational pressure drop and fluid inflow effects.

This paper presents a comprehensive, transient, semianalytical model for predicting performance of horizontal wells. By using the principles of superposition in space and time and by using mass balance equations in the reservoir and the wellbore, we solve the problem in an implicit manner.

7.2 Reservoir Model

The well is divided into segments and into each segment flow from the reservoir is described using the transient, three-dimensional, uniform flux model of Babu and Odeh⁷. Babu and Odeh have developed a general solution for the productivity of a horizontal well placed in a box-shaped drainage volume with no flow boundaries and uniform flux as the boundary condition at the well. As shown in Figure 7.1, the reservoir contains a horizontal well of radius r_w and length L . All six external boundaries of the drainage volume are of the no-flow type. The drainage volume has a height h , length a , and width b . As shown in the figure, the well extends from (x_0, y_1, z_0) to (x_0, y_2, z_0) and is parallel to the y -axis. The well can be placed anywhere in the drainage volume but must be parallel to one of the coordinate axes and can be of any length ($L \leq b$). The reservoir should be homogeneous but can be anisotropic. Porosity ϕ is constant. The reservoir permeabilities in the x , y and z directions are k_x , k_y , and k_z . The reservoir has a single-phase, slightly compressible fluid. Initially at time $t=0$, the pressure is uniform throughout the reservoir and is equal to P_{ini} . At time $t=0^+$, fluid is withdrawn from the well at a constant rate of q . Under these circumstances, they have developed a solution to calculate pressure at any time ($t > 0$), and at any position ($0 \leq x \leq a, 0 \leq y \leq b, 0 \leq z \leq h$) in the system. More details of this model can be found in their paper⁷. Babu and Odeh's uniform flux solution was given as:

$$\Delta P \equiv P_{ini} - P(x, y, z, t) = \left[\frac{886.9 B \mu q}{a b h L \alpha} \right] \int_0^{y_2} \int_{y_1}^{y_2} (S_1 \cdot S_2 \cdot S_3) dy_0 d\lambda \quad (7.1)$$

where S_1 , S_2 , and S_3 are instantaneous point sink functions (Green's functions) located at (x_0, y, z_0) and satisfying no-flow boundary conditions at $x=0, a$; $y=0, b$; and $z=0, h$. The x , y and z coordinates at the center of the well are x_0 , y_0 , and z_0 . The uniform flux solution of Babu and Odeh, in Eq. 7.1, can be written as,

$$\Delta P = (P_{ini} - P) = q \cdot F(t) \quad (7.2)$$

The above equation, along with the principle of superposition in space and time, is used to describe three-dimensional reservoir flow in the following development of models for horizontal wells.

7.3 Infinite-Conductivity Well Model

Babu and Odeh's model assumes flux along the well to be uniform. But the flux can be uniform only if the well is fully penetrating in the reservoir, resulting in two-dimensional flow. When the well is partially penetrating, the pressure support from the reservoir at the two ends of the well causes higher flux at the ends of the well as compared to the center of the well. For cases where the pressure drop in the wellbore is not very significant compared to the drawdown, it is far better to assume that the wellbore is infinitely conductive rather than to make the assumption of uniform flux along the well.

The principle of superposition in space and time is used to accomplish this task. The original well is divided into n_{seg} segments. Well segment 1 is assumed to lie closest to the heel of the well, while segment n_{seg} lies closest to the toe of the well. The uniform flux solution as given by Eq. 7.2 is applied for each segment. The interaction between segments is taken into account by the superposition of pressures in space. The pressure node for every segment is placed at the center of that segment and radially at a distance r_w from the center of the well. So the nodal pressure represents the reservoir pressure on the outer surface of the wellbore that is in contact with the reservoir. At time Δt , we can write:

$$\Delta P_1 = q_1 F_{1,1}(\Delta t) + q_2 F_{1,2}(\Delta t) + \dots + q_{n_{seg}} F_{1,n_{seg}}(\Delta t) \quad (7.3)$$

where $\Delta P_1 = (P_{ini} - P(x, y, z))$ and gives the pressure drop at the pressure node of well segment 1 due to its own flow and flow from other well segments. The location (x, y, z) is a point on the well circumference. The term $F_{m,n}$ in the above equation stands for the effect of well segment n on well segment m . Here m is the segment under consideration.

Similar equations can be written for all other segments. For example, for well segment m ,

$$\Delta P_m = q_1 F_{m,1}(\Delta t) + q_2 F_{m,2}(\Delta t) + \dots + q_{n_{seg}} F_{m,n_{seg}}(\Delta t) \quad (7.4)$$

The unknowns in this problem are n_{seg} number of q 's at the nodes of the n_{seg} segments. A total of n_{seg} equations are needed to obtain a unique solution to the system of equations. By taking the pressure difference between node 1 and every other node, we can form $(n_{seg}-1)$ equations. For example, the pressure difference between node 1 and node m is given as:

$$\Delta P_i - \Delta P_m = P_m - P_i = q_1 [F_{1,1}(\Delta t) - F_{m,1}(\Delta t)] + q_2 [F_{1,2}(\Delta t) - F_{m,2}(\Delta t)] + \dots + q_{n_{seg}} [F_{1,n_{seg}}(\Delta t) - F_{m,n_{seg}}(\Delta t)] \quad (7.5)$$

Because of the infinite conductivity assumption, pressure is constant in the wellbore. Hence the term on the left-hand side of Eq. 7.5 is equal to zero. At this point only one more equation is needed to close the system of equations. Depending on the well constraint, it can be either total flow rate specified or bottom-hole pressure specified. If the total rate Q is specified, then the last equation will be

$$Q = q_1 + q_2 + \dots + q_{n_{seg}} \quad (7.6)$$

If the bottom-hole pressure is specified, then the last equation will be

$$\Delta P_i = P_{ini} - P_i = q_1 F_{1,1}(\Delta t) + q_2 F_{1,2}(\Delta t) + \dots + q_{n_{seg}} F_{1,n_{seg}}(\Delta t) \quad (7.7)$$

where P_i is the pressure at the node of segment 1 and is equal to the specified bottom-hole pressure. Solving the system of n_{seg} equations given above, the q 's from each node for the first time step can be calculated. By applying superposition in time, this solution can be marched through time. It is assumed that all time steps are of equal length and are equal to Δt . Therefore, when $t = 3\Delta t$ (at the end of the third time step), we have

For well segment 1:

$$\Delta P_1 = q_1 F_{1,1}(3\Delta t) + \Delta_2 q_1 F_{1,1}(2\Delta t) + \Delta_3 q_1 F_{1,1}(\Delta t) + q_2 F_{1,2}(3\Delta t) + \Delta_2 q_2 F_{1,2}(2\Delta t) + \Delta_3 q_2 F_{1,2}(\Delta t) + \dots + q_{n_{seg}} F_{1,n_{seg}}(3\Delta t) + \Delta_2 q_{n_{seg}} F_{1,n_{seg}}(2\Delta t) + \Delta_3 q_{n_{seg}} F_{1,n_{seg}}(\Delta t) \quad (7.8)$$

For well segment 2:

$$\begin{aligned}
\Delta P_2 &= q_1 F_{2,1}(3\Delta t) + \Delta_2 q_1 F_{2,1}(2\Delta t) + \Delta_3 q_1 F_{2,1}(\Delta t) \\
&+ q_2 F_{2,2}(3\Delta t) + \Delta_2 q_2 F_{2,2}(2\Delta t) + \Delta_3 q_2 F_{2,2}(\Delta t) \\
&+ \dots + q_{n_{seg}} F_{2,n_{seg}}(3\Delta t) + \Delta_2 q_{n_{seg}} F_{2,n_{seg}}(2\Delta t) + \Delta_3 q_{n_{seg}} F_{2,n_{seg}}(\Delta t)
\end{aligned} \tag{7.9}$$

and for well segment n_{seg} :

$$\begin{aligned}
\Delta P_{n_{seg}} &= q_1 F_{n_{seg},1}(3\Delta t) + \Delta_2 q_1 F_{n_{seg},1}(2\Delta t) + \Delta_3 q_1 F_{n_{seg},1}(\Delta t) \\
&+ q_2 F_{n_{seg},2}(3\Delta t) + \Delta_2 q_2 F_{n_{seg},2}(2\Delta t) + \Delta_3 q_2 F_{n_{seg},2}(\Delta t) \\
&+ \dots + q_{n_{seg}} F_{n_{seg},n_{seg}}(3\Delta t) + \Delta_2 q_{n_{seg}} F_{n_{seg},n_{seg}}(2\Delta t) + \Delta_3 q_{n_{seg}} F_{n_{seg},n_{seg}}(\Delta t)
\end{aligned} \tag{7.10}$$

In the above array of equations, q_1 is the flow rate from the 1st segment at $t = \Delta t$; $\Delta_2 q_1 = q_1(2\Delta t) - q_1(\Delta t)$ and $\Delta_3 q_1 = q_1(3\Delta t) - q_1(2\Delta t)$. The equation for well segment 1 at the n^{th} time step can be written in expanded form as,

$$\begin{aligned}
[P_{wf} - P_i(n\Delta t)] &- [q_1(\Delta t)F_{1,1}(n\Delta t) + (q_1(2\Delta t) - q_1(\Delta t))F_{1,1}((n-1)\Delta t) \\
&+ (q_1(3\Delta t) - q_1(2\Delta t))F_{1,1}((n-2)\Delta t) + \\
&\dots + (q_1(n\Delta t) - q_1((n-1)\Delta t))F_{1,1}(\Delta t)] \\
&- \dots - [q_{n_{seg}}(\Delta t)F_{1,n_{seg}}(n\Delta t) + (q_{n_{seg}}(2\Delta t) - q_{n_{seg}}(\Delta t))F_{1,n_{seg}}((n-1)\Delta t) \\
&+ (q_{n_{seg}}(3\Delta t) - q_{n_{seg}}(2\Delta t))F_{1,n_{seg}}((n-2)\Delta t) \\
&+ \dots + (q_{n_{seg}}(n\Delta t) - q_{n_{seg}}((n-1)\Delta t))F_{1,n_{seg}}(\Delta t)] = 0
\end{aligned} \tag{7.11}$$

In these equations, F 's, are defined by Eq. 7.2. $F_{m,n}$ in the above equation stands for the effect of well segment n on well segment m . The well constraint equation for this set is obtained based on the type of well constraint that is active at that time. Solving the above set of equations, as was done for the first time step, can yield the flow rate distribution for each well segment at the third time step. This procedure can be repeated for all the time steps.

7.3.1 Example Problem. By using an example problem (Table 7.1), the results from the uniform flux well model and the infinite conductivity well model are compared. The results shown in Figure 7.2 are a snapshot of the flow after 20 days of production. Flow in the reservoir is under pseudosteady state at this time. The problem was run with a flow rate constraint. End-effects are more pronounced as the number of well segments is increased. Notice that, because of symmetry, results from the uniform flux case exactly coincide with those from the 2 well segments case.

Figure 7.3 shows the change in well bottom-hole pressure as the well is moved from uniform flux condition to infinite conductivity condition by increasing the number of well segments in the

model. The results show that the uniform flux case produces less than the infinite conductivity case. The reason for this is that in order to obtain the same flux at the center as at its ends, the uniform flux case requires extra pressure drop to force some of the fluid to come to the center which would otherwise go to the ends. The ends of the well, being exposed to a larger portion of the reservoir than the center of the well, gain access to reservoir fluids more easily than the inner portions of the well. As evident from Figure 7.3, the solution becomes more and more accurate as the number of well segments increases. For this particular case, about 32 divisions seemed to be adequate (this is of course problem-dependent).

7.3.2 Applications of Infinite Conductivity Wellbore Model. This model can be used as a reference solution for productivity calculations of horizontal wells with insignificant pressure drop in the wellbore. Being a semianalytical model, it does not suffer from numerical dispersion and grid sensitivities that may result from numerical simulation. It can also be used in a numerical simulator to calculate the well index for each grid block where a well is completed (Jasti *et al.*)⁸.

7.4 Finite-Conductivity Well Model

In this section we develop the solution for a horizontal well with a finite conductivity.

The well is again divided into several segments. For each well segment, the uniform flux solution of Babu and Odeh⁷ is used to describe flow in the reservoir. Flow in the wellbore is modeled using a well flow model that considers frictional, accelerational, and radial inflow effects at the well. These two flows are then coupled using the equations given below. The wellbore is divided into n_{seg} number of segments, as shown in Figure 7.4. Segment 1 represents the heel of the well, while segment n_{seg} represents the toe.

Then there are $4 n_{seg}$ number of unknowns in this problem;

1. Pressures at reservoir nodes: $P_1, P_2, \dots, P_{n_{seg}}$
2. Pressures at well nodes: $p_{w,1}, p_{w,2}, \dots, p_{w,n_{seg}}$
3. Flow rates from reservoir nodes to well nodes:
 $q_1, q_2, \dots, q_{n_{seg}}$
4. Flow rates in well segments: $q'_{w,1}, q'_{w,2}, \dots, q'_{w,n_{seg}}$

Mass Balance. Flow rate at any node in the wellbore can be related by mass balance to flow rate from the reservoir. Assuming that the density of fluids is constant in the wellbore, for simplicity, we can write:

$$q_{w,n} = \sum_{i=n}^{n_{seg}} q_i \quad (7.12)$$

n_{seg} number of such mass balance equations can be written for all the well nodes.

Pressure Continuity. The reservoir nodal pressure is located on the outer side of the well at a distance r_w from the center of the well. The well nodal pressure for the same segment is located at the center of the well. Since the momentum balance equations in the wellbore are one-dimensional in nature, well nodal pressure represents the well pressure for the entire well cross-section. Hence, reservoir nodal pressure can be equated to the well nodal pressure to establish pressure continuity between the reservoir and the wellbore:

$$P_n = p_{w,n} \quad (7.13)$$

n_{seg} such equations can be written.

Flow Equations. The equations that result from superposition in space of all well segments are referred to as flow equations. For example, the flow equation at $t = n\Delta t$, which is the n th time step, for the 1st well segment is given by Eq. 7.11. Similar equations can be written for all well segments. There will be n_{seg} number of these equations.

Pressure Drop Equations. The pressure at a given well node is related to the pressure at the downstream well node through the wellbore momentum balance equations. Flow in the wellbore is calculated using a pipe flow model. There are $(n_{seg} - 1)$ such equations. Frictional, accelerational, and radial inflow effects are considered. Frictional pressure drop gradient is calculated using standard equations for flow in pipes (e.g., Govier and Aziz⁹). By calculating the Reynolds number for each segment, the state of fluid — laminar, transitional or turbulent — can be determined and appropriate relationship for friction factor can be applied. Here we use Jain's¹⁰ explicit approximation of the implicit Colebrook-White friction factor relationship. To account for the effect of radial inflow, the following approach is used.

Recent studies by Ouyang *et al.*¹¹ have demonstrated that for radial inflow the friction factor must be modified as follows:

For laminar flow,

$$f = \frac{64}{N_{Re}} \left[1 + 0.04304 N_{Re,w}^{0.6142} \right] \quad (7.14)$$

and for turbulent flow,

$$f = f_0 \left[1 - 0.0153 N_{Re,w}^{0.3978} \right] \quad (7.15)$$

where f_0 is the friction factor calculated when there is no fluid inflow through the wall and $N_{Re,w} = q_s \rho / \pi \mu$ is the inflow Reynolds number. The variable q_s is the inflow rate per unit length of the well. Note that the effective friction factor due to inflow increases if the flow is laminar and decreases when the flow is turbulent.

Accelerational Effects. Figure 7.5 displays well segment n . The fluid is entering from the right of this segment and leaving from the left. Let q_A be the axial flow rate entering the pipe. Let q_I be the flow rate of the fluid entering the well segment from the reservoir. The change in momentum, F_{acc} , for this segment is:

$$F_{acc} = \rho A (v_2^2 - v_1^2) \quad (7.16)$$

with

$$v_1 = \frac{q_A}{A} \quad (7.17)$$

$$v_2 = \frac{q_A + q_I}{A} \quad (7.18)$$

Substituting Eqs. 7.17 and 7.18 in Eq. 7.16, we get an expression for the accelerational pressure drop over the segment:

$$\Delta P_{acc} = \frac{F_{acc}}{A} = \frac{\rho}{A^2} [q_I^2 + 2q_A q_I] \quad (7.19)$$

Constraint Equations.

For flow rate control:

$$\sum_{i=1}^{n_{seg}} q_i - Q_{max} = 0 \quad (7.20)$$

and for bottom-hole pressure control:

$$p_0 - p_{wf,min} = 0 \quad (7.21)$$

where Q_{max} is the maximum flow rate desired and $p_{wf,min}$ is the specified minimum bottom-hole pressure. Superposition in time allows us to change the constraint at any time. For the model with a specified flow rate, and as soon as the minimum bottom-hole pressure is reached, the bottom-hole pressure constraint is enforced and the flow rate is calculated.

As a result of pressure continuity, reservoir nodal pressures can be used in place of well nodal pressures. Furthermore, the mass balance equations indicate that the well nodal flow rates $q_{w,i}$ can be replaced by the reservoir nodal flow rates q_i using Eq. 7.12. Pressures and flow rates corresponding to well segments can be eliminated using the pressure continuity and mass balance equations so that only the reservoir pressures and flow rates are left as the unknown variables. After this reduction, there are only $2n_{seg}$ unknowns comprising of reservoir pressures ($P_1, \dots, P_{n_{seg}}$) and reservoir flow rates ($q_1, \dots, q_{n_{seg}}$). These unknowns can be calculated from the $2n_{seg}$ equations given by Eq. 7.11 and Eqs. 14 to 21. It should be noted that the radial inflow and accelerational effects calculated by Eqs. 14 to 19 are part of the wellbore pressure drop equations. This kind of reduction in variables reduces the computational time needed to solve the equations implicitly by Newton's method.

7.5 Testing the Models

These models were tested with published results. Since all the works available in the literature are less general than these models, a direct match is not possible. Instead, the model was tested by making necessary simplifications where to compare with the published results. The works Dikken¹, Landman², and Novy³ assume that flow in the reservoir is calculated using a supplied

productivity index. Our finite-conductivity model was also run with the productivity index supplied and with Blasius' formula to calculate frictional pressure drop in the wellbore as was done by Dikken and Landman. Our results were compared against those of Dikken and Landman under the same conditions. An excellent match was obtained. Results from our model were also compared with those predicted by the Eclipse¹² reservoir simulator using its wellbore friction option. Again a good match was found between our results and the results from Eclipse, both for the infinite-conductivity and finite-conductivity models. A fine grids was used in the simulator to minimize discretization errors. More details on the testing can be found in Penmatcha¹³.

Running a simulator with very fine grids is not economical under many circumstances. The advantage of analytical models over a numerical simulator is that the analytical models do not suffer from numerical dispersion, as is the case with numerical simulators when grids are not fine enough in areas of high pressure gradients. Also, a well index must be supplied to the simulator for each grid block where a well is completed. An analytical model of the type discussed here is needed to calculate the correct well index (Jasti *et al.*⁸).

7.5.1 Transient Effects

In this section we use the finite-conductivity well model described above to calculate the effects of friction under transient conditions.

Data from Table 7.1 are used. Figure 7.6 shows variation of drawdown at the heel with time for four cases: (a) wellbore pressure drop is ignored, (b) only friction considered, (c) friction and inflow effects considered, and (d) friction, inflow and acceleration considered. All of the cases are for a fixed flow rate of 10,000 STB/day. Consideration of inflow effects slightly reduces the frictional pressure drop in the wellbore due to the lubrication effect of the incoming fluid on turbulent flow in the wellbore. Accelerational effects, on the other hand, increase the pressure drop in the wellbore. But for this case, both the inflow and accelerational effects are small. Figure 7.6 also shows drawdown at the heel without pressure drop in the wellbore. It shows that pseudosteady state is established in this reservoir at around 2 days. During pseudosteady state flow, the drawdown at the heel when friction is considered is about 172% higher than the drawdown when friction is ignored.

Accelerational effects were found to be of greater significance for shorter wells and high flow rates. Details on this are presented in the Appendix. Figure 7.7 shows the results for a well with the same data as in Table 7.1 except that the well is only 1,000 ft long and is producing 30,000 STB/day. Accelerational pressure drop is a significant fraction of frictional pressure drop in this case.

If the flow rate remains constant, we would expect the flow distribution along the well to remain unchanged after pseudosteady state is reached. Figures 7.8 and 7.9 compare inflow flux calculation with and without pressure drop at 0.01 and 10 days, respectively. The abscissa in the figures denotes dimensionless length x/L , where x is the distance along the well from the heel and L is the length of the well. The ordinate in the figures denotes dimensionless inflow, $q_x L / Q$, where q_x is the inflow rate per unit length at a given location of the well and Q is the total

production rate from the well. Dimensionless inflow is defined in such a way that it is equal to one when inflow is uniform along the well. These figures show the error in flux calculations when wellbore pressure drop is neglected. The differences are higher at 0.01 days than at 10 days.

Frictional effects are worse during early transient flow having a highly skewed inflow profile along the well. As pseudosteady state is reached inflow becomes less skewed. During transient flow, the effective drainage volume in the reservoir is smaller than it is during pseudosteady state. Hence the high pressure drawdown at the well's heel, induced from the pressure drop in the wellbore, can influence most of the drainage volume and can pull most of the fluids being drained towards the heel. But during pseudosteady state, because of the larger drainage volume involved, it will be harder for the well's heel to influence most of the fluids being drained. And hence, inflow along the wellbore is more nonuniform during the transient flow than it is during pseudosteady state flow.

These results clearly show that during the early transient period, for cases with significant wellbore pressure drop, a much greater force drives the fluids to move towards the heel than towards the toe. This could cause premature movement of oil/water or gas/oil contacts towards the well. As pseudosteady state is reached the flow becomes more uniformly distributed, but the drawdown is still higher at the heel.

The above discussion indicates that ignoring wellbore pressure drop can lead to overestimations of production rate and breakthrough times. For a reservoir with high permeability, frictional effects are in general important but transient effects do not last very long; for a low permeability reservoir the transient effects may be long but the frictional effects are generally small. While frictional effects are only moderately important for a medium permeability reservoir during the pseudosteady period, they may be significant during initial stages of transient flow.

7.5.2 Pseudosteady State

Pseudosteady state flow is the predominant flow in many reservoirs. Here we analyze the effects of some wellbore parameters at 10 days, by which time the flow in the reservoir being considered has reached pseudosteady state. Data presented in Table 7.1 are used.

Pressure Profile in the Well. Figure 7.10 shows the pressure profile in the well at 10 days. Average pressure in the reservoir at this time is 3981.80 psia and pressure in the well at its toe is 3981.06 psi while pressure at its heel is 3960.95 psia. This plot shows that well pressure at the toe can be close to the average reservoir pressure, resulting in a very low drawdown at the toe. While the drawdown at the heel in this case is about 20.85 psi, the pressure drop in the wellbore is 20.11 psi. So the pressure drop in the wellbore is almost as high as the drawdown at the heel of the well, indicating that ignoring wellbore pressure drop for this case would lead to large overprediction of well productivity.

Effect of Well Diameter and Flow Rate. Figures 7.11 and 7.12 show the effect of well radius and flow rate on flow distribution along the well at a time of 10 days (pseudosteady state). As these figures show, flow becomes more skewed as the well radius is reduced or when the well production rate is increased. The distribution of inflow along the well becomes important when there is a gas cap and/or a water aquifer, because the higher drawdown at the heel will shorten breakthrough time. A numerical simulator can show the extent by which this nonuniformity of inflow along the well can affect water breakthrough time and post-breakthrough behavior. We cannot show these effects directly by using the finite conductivity well model, because this model can be used only for single phase flow. Nevertheless, the semianalytical model can be useful for obtaining an approximate idea about the consequences of nonuniform influx along the well. As expected, effects of wellbore pressure drop can be reduced by drilling large diameter wells or producing at low flow rates.

7.6 Effect of Directional Permeabilities

The finite-conductivity well model developed in this paper can also be used for anisotropic reservoirs, and the individual effects of directional permeabilities on wellbore pressure drop can be explored. Here we consider four cases:

1. Base case where all the permeabilities are equal:

$$k_x = k_y = k_z = 3000 \text{ mD}$$

2. $k_x = 3 \text{ mD}$, $k_y = 3000 \text{ mD}$, $k_z = 3000 \text{ mD}$

3. $k_x = 3000 \text{ mD}$, $k_y = 3 \text{ mD}$, $k_z = 3000 \text{ mD}$

4. $k_x = 3000 \text{ mD}$, $k_y = 3000 \text{ mD}$, $k_z = 3 \text{ mD}$

Parameters are the same as in the example problem (Table 7.1). A schematic of the well location appears in Figure 7.13. Results are shown in Table 7.4. The ratio of drawdowns with and without wellbore pressure drop is highest when all permeabilities are high. It is lowest when k_x is low because most of the flow to this well is coming from the x-direction, which is normal to the well axis. The ratio indicates the error introduced by neglecting wellbore pressure drop. This ratio reduces only slightly when permeabilities in the y- and z-directions are reduced, since not much flow is coming in from these directions.

Another interesting phenomenon takes place when k_y is very small, as in Case 3. Inflow is highly nonuniform in the beginning because of high pressure drop at the well heel, but it

stabilizes quickly as the fluid near the well heel is depleted faster than that towards the toe. Figure 7.14 shows how fluid inflow along the well changes during the first 10 days of production at a constant rate of 10,000 STB/day. It will be interesting to see how these effects are reflected in water breakthrough time and post breakthrough behavior. For this, it is necessary to use a multiphase simulator.

7.7 Constant Drawdown at Well Heel

Water will cusp into the heel of a well if the pressure gradient created by viscous forces is larger than buoyancy forces that act on oil and water because of density differences. Since the density difference between these fluids is more or less constant, so are the buoyancy forces. So water can be controlled from breaking through into a well by controlling the viscous pressure drop. This may be also necessary in order to control sand production from the well.

Our semianalytical models can also be run with constant drawdown at the heel by changing the constraint equation to:

$$\bar{P}_t - P_{heel,t} = \Delta P \quad (7.23)$$

where $\bar{P}_t$ is average pressure in the reservoir at time t , $P_{heel,t}$ is the pressure at the well heel, and ΔP is the constant drawdown desired.

An example case was run with constant drawdown of 10 psi. Drawdown is maintained at 10 psi as long as the heel pressure is above the minimum bottom-hole pressure of the well. Once the pressure at the heel reaches the minimum bottom-hole pressure, models use the constant bottom-hole pressure constraint. Figure 7.15 shows the effect of friction for this case. The models were run both for a finite-conductivity and an infinite-conductivity wellbore. For both these cases, the production rate is higher during the transient period but soon stabilizes as pseudosteady state is reached. For the infinite-conductivity case, at about 90 days, pressure at the well heel reached the minimum bottom-hole pressure and hence the flow rate started dropping. This figure shows that the flow rate from the well is only 6,000 STB/day when the wellbore pressure drop is considered. Ignoring the wellbore pressure drop, the well production rate is overpredicted by as high as 9,000 STB/day.

7.8 Running the Models with Other Constraints

Superposition in time allows both the infinite and finite conductivity well models to be run under either constant flow rate, or constant bottom-hole pressure, or under constant flow rate followed by constant bottom-hole pressure constraints. Since these constraints are the most commonly used bottom-hole conditions, the models are useful for most of the flow conditions usually encountered. The section on transient effects shows how the models can be run under constant

flow rate conditions. This section shows how the models behave when run under the other two constraints using the parameters in Table 7.1.

Figure 7.16 shows the results when the models are run under a minimum bottom-hole pressure of 3900 psi. During very early time, there is a large difference between the flow rates predicted by the finite-conductivity and infinite-conductivity well models. This shows again that wellbore pressure drop effects are high during the early transient flow period. The lines in the figure cross at about seven days because the finite conductivity model has a higher pressure support from the reservoir after this time due to its low production rates during the early time. Figure 7.17 shows the results when the models are initially run under a constant flow rate of 10,000 STB/day and then switched to a bottom-hole pressure constraint when the pressure at the well heel reaches 3700 psi. The constant flow rate plateau period is different for the infinite-conductivity and finite-conductivity cases.

Conclusions

1. Flexible, comprehensive, three-dimensional, transient semianalytical models are presented for infinite-conductivity and finite-conductivity wellbores in isotropic or anisotropic reservoirs.
2. The assumption of uniform flux leads to lower productivity than the case of infinite conductivity.
3. The finite-conductivity well model presented in this paper takes frictional, accelerational, and fluid inflow effects into account.
4. Ignoring wellbore pressure drop can result in the overprediction of well productivity, and may be lead to overestimation of water/gas breakthrough times.
5. Inflow into the well is more nonuniform under early transient conditions than it is under pseudosteady state conditions.
6. The semianalytical models presented here can be used to provide reference solutions to check the accuracy of numerical simulators.
7. The models presented can be run under various operating constraints.

Nomenclature

- a = reservoir length, ft
- A = well cross-sectional area, ft²
- b = reservoir width, ft
- C = constant
- d = well diameter, ft

f = Moody's friction factor
 f_0 = friction factor with no fluid inflow into the well
 F = a function, defined in Eq. 7.2
 h = reservoir height, ft
 k = permeability, mD
 L = well length, ft
 n_{seg} = number of well segments
 N_{Re} = Reynolds number
 $N_{Re,w}$ = wall-Reynolds number
 P = reservoir pressure, psi
 $\bar{P}$ = average reservoir pressure, psi
 p = well pressure, psi
 Q = total well production rate, STB/day
 q = flow rate of a well segment, STB/day
 r = radius, ft
 S = point sink function
 t = time, days
 v = fluid velocity, ft/sec
 x = dimension, ft
 y = dimension, ft
 z = dimension, ft

Greek Symbols

μ = oil viscosity, cp
 ρ = density of oil, lbm/ft³

Subscripts

acc = accelerational
 A = axial
 $heel$ = heel of the well
 i = well segment
 ini = initial

I = inflow
 max = maximum
 min = minimum
 s = specific
 w = well

Acknowledgments

We would like to thank Dr. D.K. Babu, Mobil for valuable discussions on his analytical model. This work was supported by the US department of Energy through Contract Number DF-FG22-93BC14862. Support was also provided by the Stanford University Horizontal Well Affiliate Program.

References

1. Dikken, B. J.: "Pressure Drop in Horizontal Wells and its Effects on Production Performance," *{JPT}*, 1426-1433 (November 1990).
2. Landman, M. J.: "Analytical Modeling of Selectively Perforated Horizontal Wells," *{Journal of Petroleum Science & Engineering}*, 10, 179-188 (1994).
3. Novy, R. A.: "Pressure Drop in Horizontal Wells: When Can They be Ignored?," *{SPE Reservoir Engineering}*, 29-35 (February 1995).
4. Penmatcha, V. R., Arbabi, S., and Aziz, K.: "Effects of Pressure Drop in Horizontal Wells and Optimum Well Length," paper SPE 37494, presented at the SPE Production Operations Symposium, Oklahoma City, Oklahoma (March 9-11 1997).
5. Ozkan, E. Sarica, C., and Hacıislamoglu, M.: "Effect of Conductivity on Horizontal Well Pressure Behavior," *{SPE Advanced Technical Series}*, Vol. 3, No. 1, 85-93 (1995).
6. Sarica, C., Hacıislamoglu, M., Raghavan, R., and Brill, J. P.: "Influence of Wellbore Hydraulics on Pressure Behavior and Productivity of Horizontal Gas Wells," paper SPE 28486, presented at the SPE 69th Annual Technical Conference and Exhibition held in New Orleans, Louisiana (September 1994).
7. Babu, D. K., and Odeh, A. S.: "Productivity of a Horizontal Well," *{SPERE}*, 417-421 (November 1989).
8. Jasti, J. K., Penmatcha, V. R., and Babu, D. K.: "Use of Analytical Solutions in Improvement of Simulator Accuracy," paper SPE 38887, presented at the SPE Technical Conference and Exhibition, San Antonio, Texas (October 5-8 1997).
9. Govier, G. W., and Aziz, K.: *The Flow of Complex Mixtures in Pipes*, Van Nostrand Reinhold (1972).

10. Jain, A. K.: "Accurate Explicit Equation for Friction Factor," *Journal of Hydraulics Div. ASCE* (1976) 102, 674-677.
11. Ouyang, L-B., Arbabi, S., and Aziz, K.: "General Wellbore Flow Model for Horizontal, Vertical, and Slanted Well Completions," paper SPE 36608, presented at the SPE Annual Technical Conference and Exhibition, Denver, Colorado (October 6-9 1996).
12. Eclipse 1996A, Wellbore Friction Option, Geoquest, Schlumberger.
13. Penmatcha, V. R.: "Modeling of Horizontal Wells with Pressure Drop in the Well," Ph.D. thesis at Stanford University, California, USA (1997).

Appendix: Accelerational vs. Frictional Pressure Drop

The analysis presented here compares the total accelerational pressure drop in the wellbore to the total frictional pressure drop. Most works in the literature that have coupled flow in the wellbore and the reservoir consider only frictional pressure drop. Some of them (Novy, 1995) claim that they did not consider accelerational pressure drop in the wellbore because it was very small compared to the frictional pressure drop. The analysis presented below shows which factors can make accelerational pressure drop significant.

To simplify the analysis, we assume that Blasius' formula works well for calculating the frictional pressure drop. Using Eq. 7.19 and considering the entire length of the well as the control volume in Figure 7.5, total accelerational pressure drop in the wellbore is given as:

$$\Delta p_{acc, total} = \frac{\rho}{A^2} Q^2 = M \frac{\rho Q^2}{d^4} \quad (7.A1)$$

where M is a constant. The above equation indicates that the total accelerational pressure drop in a well is independent of the length of the well and the influx distribution along the well. This makes good sense because accelerational pressure drop is caused by the total change in kinetic energy of the fluid. Blasius' formula is given as:

$$f = C_b N_{Re}^{-\alpha} \quad (7.A2)$$

where C_b and α are constants. Frictional pressure gradient in a pipe can be calculated by

$$\frac{dp}{dx} = 1.079 \times 10^{-4} \frac{f \rho v^2}{d} \quad (7.A3)$$

Using Eq. 7.A3 along with Eq. 7.A2, we get the frictional pressure gradient in the well as:

$$\frac{dp_{fric}(x)}{dx} = Cq(x)^{2-\alpha} \rho^{1-\alpha} \frac{\mu^\alpha}{d^{5-\alpha}} \quad (7.A4)$$

where C is a constant.

Total frictional pressure drop in the well depends on the distribution of inflow along the well. Two different inflow distributions are considered in this analysis. The first inflow distribution considered assumes that all the inflow from the reservoir to the wellbore is occurring at the well's toe. Total frictional pressure drop in the well reaches a maximum when all the flow is entering the well at its toe. This case may not be very practical but will serve well for the analysis of this problem by looking at one extreme. Under this situation, we get

$$\Delta p_{fric,total} = CQ^{2-\alpha} \rho^{1-\alpha} \frac{\mu^\alpha}{d^{5-\alpha}} L \quad (7.A5)$$

where $\Delta p_{fric,total}$ is the total frictional pressure drop in the well and L is the length of the wellbore.

Dividing Eq. 7.A1 with Eq. 7.A5 results in the ratio of total accelerational to frictional pressure drop in the well, and is given as:

$$\frac{\Delta p_{acc,total}}{\Delta p_{fric,total}} = \left(\frac{M}{C} \right) \frac{\rho^\alpha Q^\alpha d^{1-\alpha}}{\mu^\alpha L} \quad (7.A6)$$

The Blasius' constant α is equal to 0.25 for a smooth pipe and it approaches zero for an extremely rough pipe. Hence this constant is positive and non-zero for most of the cases. Eq. 7.A6 then indicates that the significance of the accelerational pressure drop in the wellbore as compared to frictional pressure drop increases with higher flow rates, shorter wells, large diameter wells, large fluid density or with low fluid viscosities.

The second inflow distribution considered assumes that inflow is uniform along the well. Under these circumstances, total frictional pressure drop in the wellbore is given as:

$$\begin{aligned}\Delta p_{fric,total} &= C \frac{\rho^{1-\alpha} \mu^\alpha}{d^{5-\alpha}} \int_0^L \left(\frac{Qx}{L} \right)^{2-\alpha} dx \\ &= C Q^{2-\alpha} \rho^{1-\alpha} \frac{\mu^\alpha}{d^{5-\alpha} (3-\alpha)} L\end{aligned}\tag{7.A7}$$

Note that flow rate in the well at any location for this case is equal to Qx/L , with $x=0$ at the well's toe and $x=L$ at the well's heel. The ratio of the total accelerational pressure drop to the total frictional pressure drop for this case is:

$$\frac{\Delta p_{acc,total}}{\Delta p_{fric,total}} = \left(\frac{M(3-\alpha)}{C} \right) \frac{\rho^\alpha Q^\alpha d^{1-\alpha}}{\mu^\alpha L}\tag{7.A8}$$

Except for the multiplication by a constant, $(3-\alpha)$, the ratio of pressure drops still remains the same compared to Eq. 7.A6 indicating that the change in flow rate distribution did not influence the general relationship.

SI Metric Conversion Factor

bbl x 1.589873	E-01 = m ³
cp x 1.0*	E-03 = Pa.s
ft x 3.048*	E-01 = m
mD x 9.869 233	E-04 = μm ²
psi x 6.894 757	E+00 = kPa

*conversion factor is exact.

Table 7.1: Example Problem

Length of the drainage volume	6,000 ft
Width of the drainage volume	12,000 ft
Height of the drainage volume	50 ft
Permeability in x-direction	3,000 mD
Permeability in y-direction	3,000 mD
Permeability in z-direction	3,000 mD
Porosity	0.3
Reservoir initial pressure	4,000 psi
Total compressibility	$3 \times 10^{-5} \text{ psi}^{-1}$
Formation volume factor	1.05 RB/STB
Oil viscosity	1 cp
Oil density at reservoir conditions	60 lbm/ft ³
Maximum oil rate	10,000 STB/day
Minimum bottom-hole pressure	1200 psi
Well location in x-direction	3,000 ft
Heel location in y-direction	3,000 ft
Toe location in y-direction	9,000 ft
Well location in z-direction	25 ft
Well diameter	4 inch
Relative wellbore roughness	0.0005

Table 7.2: Fine Grid Distribution Used in Eclipse

Direction	Grid Distribution (ft)
x	2*1331.5 250 30 6 51*2 6 30 250 2*1331.5
y	50*45 50*15 40*150 50*15 50*45
z	25*2

Table 7.3: Coarse Grid Distribution Used in Eclipse

Direction	Grid Distribution (ft)
x	9*666.67
y	80*150
z	3*16.67

Table 7.4: Effect of Directional Permeabilities on Wellbore Pressure Drop

Case #	Drawdown	Drawdown	Ratio
1	6.69	20.85	3.107
2	523.60	550.00	1.050
3	22.79	51.60	2.264
4	21.39	44.59	2.085

Figure 7.1: Horizontal Well in a Box-shaped Drainage Volume (Babu and Odeh⁷)

Figure 7.2: Infinite Conductivity and Uniform Flux Solutions at 20 Days (Pseudosteady state) for Example in Table 7.1

Figure 7.3: Influence of the Number of Well Segments on Productivity

Figure 7.4: Well Divided into Several Segments

Figure 7.5: Control Volume to Illustrate Accelerational Effects

Figure 7.6: Performance of a Well with and without Pressure Drop in the Well
(Table 7.1 Data)

Figure 7.7: Accelerational Effects for a 1,000 ft Well Producing 30,000 STB/day.

Figure 7.8: Flux Along the well under Transient Conditions at 0.01 days (Table 7.1 Data)

Figure 7.9: Flux along the Well under Pseudosteady State Conditions at 10 days (Table 7.1 Data)

Figure 7.10: Pressure Profile along the Well under Pseudosteady State Conditions at 10 days

Figure 7.11: Flux along the Well for Different Well Diameters under Pseudosteady State Conditions

Figure 7.12: Flux along the Well for Different Flow Rates under Pseudosteady State Conditions

Figure 7.13: A Schematic of Well Location in the Reservoir

Figure 7.14: Flux Variation along the Well due to Lowered Permeability in the Well Direction

Figure 7.15: Effect of Wellbore Pressure Drop on Results with a Constant Drawdown Specified at the Heel

Figure 7.16: Results with a Minimum Bottom-hole Pressure Specified

Figure 7.17: Results with a Constant Flow Rate Followed by a Minimum Bottom-hole Pressure