Fall 2013

Computational fluid dynamics flow comparison between openhole sleeve and plug-and-perf completion in a hydraulic fractured horizontal well

Viriyah Theppornprapakorn

Follow this and additional works at: http://scholarsmine.mst.edu/masters_theses

Part of the Petroleum Engineering Commons

Department: Geosciences and Geological and Petroleum Engineering

Recommended Citation
COMPUTATIONAL FLUID DYNAMICS FLOW COMPARISON BETWEEN OPENHOLE SLEEVE AND PLUG-AND-PERF COMPLETION IN A HYDRAULIC FRACTURED HORIZONTAL WELL

by

VIRIYAH THEPPORNPRAPAKORN

A THESIS
Presented to the Faculty of the Graduate School of the MISSOURI UNIVERSITY OF SCIENCE AND TECHNOLOGY In Partial Fulfillment of the Requirements for the Degree MASTER OF SCIENCE IN PETROLEUM ENGINEERING 2013

Approved by

Dr. Shari Dunn-Norman, Advisor Dr. Ralph Flori Dr. Mingzhen Wei
Multi-stage hydraulic fracture completions in horizontal wells have facilitated economic development of natural gas from unconventional and tight gas reservoirs. Industry has relied on two horizontal multi-stage completion technologies: cemented and perforated liner or Plug-and-perf method and Openhole Sleeve Multi-stage method. Each method has its own advantages and disadvantages. However, one of the most important questions is which method yields better gas production?

Computation fluid dynamics (CFD) is a powerful tool for solving complex fluid flow problems. In well completion design, CFD plays a significant role aiding the completion engineers on making decision on well completion methods.

In this study, a three-dimensional CFD model of a horizontal well with a single transverse fracture has been constructed to compare Plug-and-perf versus Openhole sleeve multi-stage completion methods using steady state natural gas production and no formation damage. The results are shown in a relationship of dimensionless fracture conductivity and fold of increase. Parametric studies have been performed varying horizontal permeability ratio, penetration ratio and propped fracture width.

Considering only the effect from the completion method and without the presence of the natural fractures, results of this study indicate a well completed with Openhole Sleeve Multi-stage method does produce more than a well completed with the Plug-and-perf method, but the difference in production is substantially less than previous publications indicate. The results from this study were compared with the previous study using the two-dimensional models by Augustine (2011).
ACKNOWLEDGMENTS

I would like to thank Dr. Shari Dunn-Norman, my advisor, for accepting me as her graduate student. I am really grateful for her guidance and financial support throughout this study, which is the first research project under CFD group of Petroleum Engineering program.

In additions, I am so thankful to Dr. Ralph Flori and Dr. Mingzhen Wei, for serving as my thesis committees.

I would like to thank Chatetha Chumkratoke for being my friend and mentor in my time studying in Petroleum Engineering program of Missouri S&T.

I would like to thank Jinlin Zhang and Mustafa Al-Alwani from CFD group for your time listening through my practicing presentation.

Lastly, I would like to thank my mother, father, aunt and sister at my hometown, Bangkok, Thailand, for their financial and mental supports. Most of all, their love and encouragement have helped me through my hard time being away from home.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>iii</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>iv</td>
</tr>
<tr>
<td>LIST OF ILLUSTRATIONS</td>
<td>ix</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xii</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>xiii</td>
</tr>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1. UNCONVENTIONAL RESERVOIRS: TIGHT GAS AND SHALE GAS</td>
<td>2</td>
</tr>
<tr>
<td>1.1.1. Tight Gas</td>
<td>2</td>
</tr>
<tr>
<td>1.1.2. Shale Gas</td>
<td>3</td>
</tr>
<tr>
<td>1.2. FRACTURE DIRECTION AND WELL ORIENTATION</td>
<td>5</td>
</tr>
<tr>
<td>1.2.1. Principal Stresses</td>
<td>5</td>
</tr>
<tr>
<td>1.2.2. Fracture Direction</td>
<td>6</td>
</tr>
<tr>
<td>1.2.3. Well Orientation</td>
<td>6</td>
</tr>
<tr>
<td>1.2.4. Well Performance from Different Types of Fractures.</td>
<td>7</td>
</tr>
<tr>
<td>1.3. HORIZONTAL COMPLETION</td>
<td>8</td>
</tr>
<tr>
<td>1.3.1. Plug-and-perf Method (P-n-P)</td>
<td>8</td>
</tr>
<tr>
<td>1.3.2. Open Hole Multi-stage System</td>
<td>9</td>
</tr>
<tr>
<td>1.4. FRACTURE PRODUCTIVITY</td>
<td>11</td>
</tr>
<tr>
<td>1.4.1. Concept of Productivity Index</td>
<td>11</td>
</tr>
<tr>
<td>1.4.2. Notations of Hydraulic Fracture</td>
<td>12</td>
</tr>
<tr>
<td>1.4.3. Improvement of Productivity Index</td>
<td>13</td>
</tr>
<tr>
<td>1.4.3.1 Prats’ method</td>
<td>14</td>
</tr>
<tr>
<td>1.4.3.2 McGuire-Sikora chart</td>
<td>16</td>
</tr>
<tr>
<td>1.5. COMPUTATIONAL FLUID DYNAMICS IN WELL COMPLETION</td>
<td>17</td>
</tr>
<tr>
<td>1.6. THESIS OBJECTIVE</td>
<td>18</td>
</tr>
<tr>
<td>2. LITERATURE REVIEW</td>
<td>20</td>
</tr>
<tr>
<td>2.1. FIELD OBSERVATION</td>
<td>20</td>
</tr>
</tbody>
</table>
2.2. ANALYTICAL METHOD ................................................................. 22
3. CONCEPT OF STUDY .................................................................. 28
  3.1. RESERVOIR SELECTION ......................................................... 28
  3.2. CONCEPTUAL MODEL ............................................................. 28
      3.2.1. Concept of Un-stimulated Model ................................. 29
      3.2.2. Concept of Stimulated Well ........................................ 30
4. CFD AND FLUENT ................................................................. 31
  4.2. GOVERNING EQUATIONS IN CFD ........................................... 32
      4.2.1. Finite Control Volume ................................................. 32
      4.2.2. Continuity Equation ..................................................... 33
      4.2.3. Momentum Equations .................................................. 34
      4.2.4. Navier-Stokes Equations ............................................. 36
      4.2.5. Porous Media Terms in Momentum Equations ............... 36
      4.2.6. Equation Of States ....................................................... 38
      4.2.7. Energy Equation .......................................................... 38
  4.3. DISCRETIZATION SCHEME: FINITE VOLUME METHOD ............ 39
4.4. CFD SOLVER ....................................................................... 41
      4.4.1.1 Pressure-based solver .............................................. 41
      4.4.1.2 Density-based solver ................................................. 41
4.5. ANSYS-WORKBENCH AND FLUENT ..................................... 42
      4.5.1. Overview ................................................................. 42
      4.5.2. Components in FLUENT ............................................ 43
          4.5.2.1 DesignModeler ............................................... 44
          4.5.2.2 Meshing ............................................................. 45
          4.5.2.3 Setup & Solution ............................................... 47
          4.5.2.4 CFD-Post ......................................................... 49
      4.5.3. Workflow in FLUENT ............................................... 49
5. HORIZONTAL WELL MODEL VALIDATION ..................................... 51
  5.1. HORIZONTAL WELL PRODUCTION ...................................... 51
      5.1.1. Horizontal Well Equation for the Incompressible Fluid .... 51
      5.1.2. Horizontal Well Equation in Gas Reservoir ................. 52
5.2. BASIC MODEL DATA .......................................................... 52
  5.2.1. Reservoir and Natural Gas Data..................................... 52
  5.2.2. First FLUENT Horizontal Well Model............................ 53
5.3. MODEL VALIDATION .......................................................... 55
  5.3.1. Incompressible Fluid Model Validation ......................... 56
    5.3.1.1 Validation steps in the incompressible fluid model ....... 56
    5.3.1.2 Inflow performance relationship (IPR) 60
    5.3.1.3 Incompressible fluid model results ....................... 61
    5.3.1.4 New-proposed model ......................................... 63
  5.3.2. Compressible Fluid Model Validation ............................ 66
    5.3.2.1 Natural gas properties ...................................... 66
    5.3.2.2 Validation steps in the compressible fluid model ........ 72
    5.3.2.3 Compressible fluid model results ........................ 76
6. COMPLETION MODELS AND PROCEDURES .................................. 79
  6.1. FRACTURE MODEL ........................................................ 79
  6.2. PLUG-AND-PERF COMPLETION MODEL ................................ 80
  6.3. OPENHOLE MULTISTAGE COMPLETION MODEL ....................... 83
  6.4. SIMULATION AND PARAMETRIC STUDIES ............................. 85
    6.4.1. Base Case ...................................................... 86
    6.4.2. Parametric Studies .......................................... 86
      6.4.2.1 Parametric study case 1: fracture width (w) ........... 86
      6.4.2.2 Parametric study case 2: penetration ratio (x_f/r_e) .. 86
      6.4.2.3 Parametric study case 3: (k_v/k_h) ..................... 87
7. SIMULATION RESULTS ....................................................... 88
  7.1. BASE CASE RESULTS .................................................. 88
  7.2. RESULTS FROM PARAMETRIC STUDIES ............................. 89
    7.2.1. Results from the Parametric Study Case 1: Fracture Width (w) .... 89
    7.2.2. Results from the Parametric Study Case 2: Penetration Ratio (x_f/r_e) .... 92
    7.2.1. Results from the Parametric Study Case 3: k_v/k_h ............ 94
8. RESULTS DISCUSSION ...................................................... 98
  8.1. BASE CASE DISCUSSION ............................................. 98
8.1.1. Contour Analysis ........................................................................................................98
  8.1.1.1 The velocity contours of OHMS model .........................................................99
  8.1.1.2 The velocity contours of P-n-P model .........................................................101

8.1.2. Comparison of Base Case with Augustine’s Results. 104

8.2. DISCUSSION ON PARAMETRIC STUDIES ......................................................... 107
  8.2.1. Discussion on Parametric Study Case 1: Fracture Width .............................. 107
  8.2.2. Discussion on Parametric Study Case 2: Penetration Ratio .......................... 107
  8.2.3. Discussion on Parametric Study Case 3: \( k_v/k_h \) ............................................... 109

9. SUMMARY AND CONCLUSIONS ............................................................................. 113

10. FUTURE WORK ......................................................................................................... 115

APPENDICES
  A. GAS PROPERTIES .................................................................................................. 116
  B. CFD SIMULATION TUTORIAL ............................................................................. 120
  C. RAW DATA AND DIGITIZED DATA ..................................................................... 138

BIBLIOGRAPHY ............................................................................................................ 148

VITA ............................................................................................................................... 152
# LIST OF ILLUSTRATIONS

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Permeability range of formations and where required hydraulic fracturing (King, 2012)</td>
</tr>
<tr>
<td>1.2</td>
<td>Orthogonal principle stresses</td>
</tr>
<tr>
<td>1.3</td>
<td>Fracturing propagation perpendicular to the minimum horizontal stress (Bachman et al., 2007)</td>
</tr>
<tr>
<td>1.4</td>
<td>Longitudinal (left) and transverse fractures (right) in horizontal wells (Bahrami et al., 2013)</td>
</tr>
<tr>
<td>1.5</td>
<td>Plug-and-perf method (Soliman et al., 2012)</td>
</tr>
<tr>
<td>1.6</td>
<td>Open Hole Multi-stage System (Snyder et al., 2011)</td>
</tr>
<tr>
<td>1.7</td>
<td>Notations of hydraulic fracture</td>
</tr>
<tr>
<td>1.8</td>
<td>Prat’s curve (Economides et al., 1994 retrieved from Prats (1961))</td>
</tr>
<tr>
<td>1.9</td>
<td>McGuire-Sikora chart (Cholet, 2008)</td>
</tr>
<tr>
<td>2.1</td>
<td>Half of edge-drive reservoir model (Augustine, 2011)</td>
</tr>
<tr>
<td>2.2</td>
<td>1/4 of two-dimensional reservoir model (Augustine, 2011)</td>
</tr>
<tr>
<td>2.3</td>
<td>Example of Augustine’s results (Augustine, 2011)</td>
</tr>
<tr>
<td>3.1</td>
<td>Conceptual model of the un-stimulated well</td>
</tr>
<tr>
<td>3.2</td>
<td>Conceptual model of the un-stimulated well</td>
</tr>
<tr>
<td>4.1</td>
<td>Finite control volume in fixed space (Modified from Anderson, 1995)</td>
</tr>
<tr>
<td>4.2</td>
<td>Mass flow in and out of control volume (Versteeg et al., 2009)</td>
</tr>
<tr>
<td>4.3</td>
<td>Stress components (Versteeg et al., 2009)</td>
</tr>
<tr>
<td>4.4</td>
<td>Control volumes with nodes Configuration (Bakker, 2002-2006)</td>
</tr>
<tr>
<td>4.5</td>
<td>ANSYS workbench</td>
</tr>
<tr>
<td>4.6</td>
<td>Analysis system of FLUENT in ANSYS workbench</td>
</tr>
<tr>
<td>4.7</td>
<td>DesignModeler module</td>
</tr>
<tr>
<td>4.8</td>
<td>Meshing module</td>
</tr>
<tr>
<td>4.9</td>
<td>Meshing cells (ANSYS Lecture 3, 2012)</td>
</tr>
<tr>
<td>4.10</td>
<td>Setup module in FLUENT</td>
</tr>
<tr>
<td>4.11</td>
<td>CFD-Post</td>
</tr>
<tr>
<td>4.12</td>
<td>Flowchart of FLUENT analysis system</td>
</tr>
<tr>
<td>5.1</td>
<td>Symmetry planes (Bakker, 2002-2006)</td>
</tr>
</tbody>
</table>
Figure 5.2. Well pressure diagram ................................................................. 54
Figure 5.3. Horizontal well model in DesignModeler .................................................... 55
Figure 5.4. Horizontal well model after Meshing ............................................................... 56
Figure 5.5. Cell zone conditions panel ........................................................................... 57
Figure 5.6. Mass conservation monitoring ...................................................................... 59
Figure 5.7. IPR example ................................................................................................. 60
Figure 5.8. IPR of gas well under Darcy’s law (Wang et al., 2009) ................................. 61
Figure 5.9. IPR Comparison between FLUENT result of edge-drive model and horizontal well equation ...................................................................................... 62
Figure 5.10. Top view concept of horizontal well by Joshi (1991) .................................... 64
Figure 5.11. New proposed horizontal well model ............................................................ 65
Figure 5.12. IPR Comparison between FLUENT result of new-proposed model and horizontal well equation ...................................................................................... 65
Figure 5.13. Z-factor chart by Standing and Katz (1942) (Wang et al., 2009) .............. 69
Figure 5.14. Multi-species setup .................................................................................... 73
Figure 5.15. Critical temperature, Critical pressure, Critical specific volume and acentric factor setup .............................................................................................. 74
Figure 5.16. Fixed values setup ....................................................................................... 75
Figure 5.17 IPR Comparison between FLUENT result and horizontal well equation: natural gas model ................................................................. 76
Figure 6.1. Position of single transverse fracture .............................................................. 79
Figure 6.2. Transverse fracture in DesignModeler ............................................................. 80
Figure 6.3. Perforation holes and fracture bodies from the –X view .................................. 81
Figure 6.4. Outlets of perforation holes .......................................................................... 82
Figure 6.5. Sectional view of wall behind cased and cemented liner ............................... 83
Figure 6.6. Comparison of mesh distribution between 1 fracture model( left) and 2 fractures model(right) ...................................................................................... 84
Figure 6.7. OHMS model .............................................................................................. 84
Figure 6.8 Sectional view of OHMS ................................................................................ 85
Figure 7.1. Base case results ............................................................................................ 88
Figure 7.2. Results: parametric study case1: fracture width; P-n-P ..................................... 89
Figure 7.3. Results: parametric study case1: fracture width; OHMS ............................... 90
Figure 7.4. Results: parametric study case1: fracture width; P-n-P &OHMS .................. 91
Figure 7.5. Results: parametric study case2: penetration ratio; P-n-P ............................. 92
Figure 7.6. Results: parametric study case2: penetration ratio; OHMS .......................... 93
Figure 7.7. Results: parametric study case2: Penetration ratio; P-n-P &OHMS .......... 94
Figure 7.8. Results: parametric study case3: k_v/k_h; P-n-P ...................................... 95
Figure 7.9. Results: parametric study case3: k_v/k_h; OHMS ..................................... 96
Figure 7.10. Results: parametric study case3: k_v/k_h; P-n-P &OHMS ....................... 97
Figure 8.1. Locations of velocity contour planes.......................................................... 98
Figure 8.2. Velocity contour at mid-sectional plane of OHMS model......................... 99
Figure 8.3. Close-up view near fracture: velocity contour at mid-sectional plane of OHMS model.......................................................... 100
Figure 8.4. Velocity contour at symmetry plane of OHMS model............................. 100
Figure 8.5. Close-up view near well area: velocity contour at symmetry plane of OHMS model.......................................................... 101
Figure 8.6. Velocity contour at mid-sectional plane (-Y view) of P-n-P model.............. 102
Figure 8.7. Close-up view near well area: velocity contour at symmetry plane of P-n-P model.......................................................... 103
Figure 8.8. Augustine results case: k_v/k_h = 0.1, L = 100 ft , L/r_e = 0.5 and I_x = 0.5 with the digitized range (Augustine, 2011) .......... 105
Figure 8.9. Comparison of base case study and Augustine’s results............................ 106
Figure 8.10. Parametric study on penetration ratio on the openhole completion model (Augustine, 2011) .......................................................... 108
Figure 8.11. Comparison of parametric study on penetration ratio between OHMS results and Augustine’s results ........................................ 109
Figure 8.12. Parametric study on k_v/k_h on the openhole and cemented completion models (Augustine, 2011).......................................................... 110
Figure 8.13. Comparison of parametric study on k_v/k_h and Augustine’s results ........ 111
LIST OF TABLES

Table 5.1. Reservoir and natural gas data................................................................. 53
Table 5.2. Errors from the edge-drive model............................................................. 63
Table 5.3. Error from the new-proposed model......................................................... 66
Table 5.4. Errors from the natural gas model ............................................................ 77
Table 8.1. Mass flow rate from fracture and openhole sections in OHMS 
and overall flow from P-n-P: 150 ft results in base case .............................. 104
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>P-n-P</td>
<td>Plug-and-perf method</td>
<td>1</td>
</tr>
<tr>
<td>OHMS</td>
<td>Open Hole Sleeve Multi-Stage System method</td>
<td>1</td>
</tr>
<tr>
<td>$\sigma_v$</td>
<td>Vertical stress</td>
<td>5</td>
</tr>
<tr>
<td>$\sigma_H$</td>
<td>Maximum horizontal stress</td>
<td>5</td>
</tr>
<tr>
<td>$\sigma_h$</td>
<td>Minimum horizontal stress</td>
<td>5</td>
</tr>
<tr>
<td>$r_e$</td>
<td>Reservoir outer boundary (ft)</td>
<td>11</td>
</tr>
<tr>
<td>$J_{oil}$</td>
<td>Productivity index in oil well (STB/D-psi)</td>
<td>12</td>
</tr>
<tr>
<td>$q_o$</td>
<td>Oil flow rate (STB/D)</td>
<td>12</td>
</tr>
<tr>
<td>$p_e$</td>
<td>Outer boundary pressure (psi)</td>
<td>12</td>
</tr>
<tr>
<td>$p_{wf}$</td>
<td>Flowing bottom hole pressure</td>
<td>12</td>
</tr>
<tr>
<td>$J$</td>
<td>Productivity index in gas well (MCF/D-psi²)</td>
<td>12</td>
</tr>
<tr>
<td>$q$</td>
<td>Gas flow rate (MSCF/D)</td>
<td>12</td>
</tr>
<tr>
<td>$x_f$</td>
<td>Fracture half-length (ft)</td>
<td>12</td>
</tr>
<tr>
<td>$w$</td>
<td>Propped fracture width (in)</td>
<td>12</td>
</tr>
<tr>
<td>$h_f$</td>
<td>Propped fracture height (ft)</td>
<td>12</td>
</tr>
<tr>
<td>$C_f$</td>
<td>Fracture conductivity(md-ft)</td>
<td>13</td>
</tr>
<tr>
<td>$C_{fd}$</td>
<td>Dimensionless fracture conductivity</td>
<td>13</td>
</tr>
<tr>
<td>$k$</td>
<td>Reservoir permeability</td>
<td>13</td>
</tr>
<tr>
<td>FOI</td>
<td>Fold of increase</td>
<td>13</td>
</tr>
<tr>
<td>$J_o$</td>
<td>Productivity index before the stimulation (same unit as J)</td>
<td>14</td>
</tr>
<tr>
<td>$a$</td>
<td>Prat’s relative capacity</td>
<td>14</td>
</tr>
<tr>
<td>$r'_wD$</td>
<td>Dimensionless effective wellbore radius</td>
<td>14</td>
</tr>
<tr>
<td>$r'_w$</td>
<td>Effective well radius</td>
<td>14</td>
</tr>
<tr>
<td>$r_w$</td>
<td>Well radius</td>
<td>14</td>
</tr>
<tr>
<td>$s_f$</td>
<td>Equivalent skin due to hydraulic fracturing</td>
<td>14</td>
</tr>
<tr>
<td>$r_e$</td>
<td>Drainage/Outer boundary radius</td>
<td>15</td>
</tr>
<tr>
<td>A</td>
<td>McGuire and Sikora’s square drainage area(acres)</td>
<td>16</td>
</tr>
</tbody>
</table>
Penetration ratio ................................................................. 16
Distance from well to the edge of drainage area(ft) .......................... 16
CFD Computational Fluid Dynamics ........................................... 17
Reservoir length (ft) by Augustine (2011) .................................. 22
Reservoir radius (ft) in Augustine (2011) ................................ 22
Reservoir pressure (psi) in Augustine (2011) .............................. 22
Reservoir height (ft) in Augustine (2011) ................................ 22
Well radius (ft) in Augustine (2011) ....................................... 22
Well pressure (psi) in Augustine (2011) ................................... 22
Oil flow rate(bbl/day) in Augustine (2011) ................................. 23
Horizontal permeability (md) in Augustine (2011) ..................... 23
Oil formation volume factor (res. bbl/ bbl) in Augustine (2011) .... 23
Oil viscosity(cp) in Augustine (2011) ..................................... 23
Equivalent length (ft) in Augustine (2011) ............................... 23
Vertical to horizontal permeability ratio in Augustine (2011) ....... 23
Wellbore length (ft) in Augustine (2011) .................................. 24
2D model fracture length (ft) in Augustine (2011) ...................... 25
2D model reservoir length (ft) in Augustine (2011) .................... 25
2D model fracture width (ft) in Augustine (2011) ....................... 25
2D model reservoir width (ft) in Augustine (2011) ..................... 25
milliDarcy ........................................................................... 25
nanoDarcy ........................................................................... 25
fracture permeability (md) redefined relative conductivity .......... 26
reservoir permeability (md) redefined relative conductivity ......... 26
propped fracture width (in) :redefined relative conductivity ........ 26
propped fracture half-length (ft) redefined relative conductivity ... 26
reservoir thickness(ft) .......................................................... 29
horizontal well length .......................................................... 29
Computer-aided design .......................................................... 31
Density of fluid .................................................................... 34
Time ................................................................................... 34
\( \delta x \) Distance in x direction
\( \delta y \) Distance in y direction
\( \delta z \) Distance in z direction
\( u \) Fluid velocity in x direction
\( v \) Fluid velocity in y direction
\( w \) Fluid velocity in z direction
\( S_m \) Mass added to the continuation phase
\( p \) Normal stress
\( \tau_{ij} \) Vicious stress in j normal to i direction
\( S_{Mx} \) Body force in the x direction
\( S_{My} \) Body force in the y direction
\( S_{Mz} \) Body force in the z direction
\( \mu \) Dynamic viscosity in Navier-Stroke equations
\( S_i \) Additional body force term in the i direction
\( \alpha \) Permeability (Porous media in CFD)
\( v_i \) Superficial velocity in the i direction
\( C_i \) Initial resistance factor in the i direction
\( |v| \) Magnitude of velocity
\( v_i \) Vector of velocity in the i direction
\( \Delta n_x \) Thickness of porous media in x direction
\( \Delta n_y \) Thickness of porous media in y direction
\( \Delta n_z \) Thickness of porous media in z direction
\( \alpha \) Under-relaxation factor
\( \phi \) Current solution variable
\( \phi_{old} \) Old solution variable
\( R^\phi \) Residual sum
\( \phi_p \) Solution variable in cell P
\( a_p \) Central coefficient
\( a_{nb} \) Influence from the neighboring cells
\( b \) Contribution of the constant part of the source term
\( \text{GUI} \) Graphic user interface
q  Surface volume flow rate ( STB/d) .......................................................... 51
kH  Horizontal absolute permeability (md), .................................................. 51
h  Reservoir thickness (ft), ........................................................................... 51
B  Formation volume factor of liquid ( res bbl/STB), .................................... 51
L  Horizontal well length ( ft) ....................................................................... 51
kV  Vertical absolute permeability (md), ....................................................... 51
r_eH  Distance to the outer boundary (ft) in Horizontal well ....................... 52
q_{gas}  Natural gas flow rate at surface (MCF/d)........................................... 52
MSCF/d  1000 Standard cubic feet per day .................................................... 52
\mu  Average gas viscosity : outer boundary and wellbore (cp) ..................... 52
\bar{Z}  Average gas Z factor : outer boundary and wellbore ........................ 52
T  Reservoir temperature(°F)...................................................................... 52
kg/s  Kilograms/second .............................................................................. 58
RVB  Reservoir barrel per day ..................................................................... 59
IPR  Inflow performance relationship .......................................................... 59
q  Production rate( IPR) ............................................................................... 60
y_i  Mole fraction of component i ................................................................. 67
n  Number of moles of component i .............................................................. 67
\Sigma n_i  Total moles in mixture .................................................................. 67
MW_g  Apparent molecular weight of natural gas ........................................... 67
MW_i  Molecular weight of component i ........................................................ 67
p  Pressure in real gas law (psi) ................................................................... 67
V  Gas volume in real gas law (ft^3) .............................................................. 67
Z  Gas deviation factor ................................................................................. 67
R  Universal gas constant ............................................................................ 67
T  Absolute temperature (R) ......................................................................... 67
°F  Degree Fahrenheit .................................................................................. 67
\rho_g  Natural gas density ( lb/ft^3) ................................................................. 67
p_{pc}  Pseudo critical pressure of natural gas (psi) ........................................ 68
p_{ci}  Pseudo critical pressure of component i (psi) ....................................... 68
T_{pc}  Pseudo critical temperature of natural gas (R) ..................................... 68
T_{ci}  Pseudo critical temperature of component i (R)............................................. 68
p_{pr}  Pseudo reduced pressure ................................................................................. 68
T_{pr}  Pseudo reduced temperature.............................................................................. 68
t  Reciprocal of pseudo reduced temperature (T_{pc}/T)............................................ 69
y  Reduced density ..................................................................................................... 69
PR EOS  Peng-Robinson equation of state ................................................................. 70
T_{cm}  Critical temperature of mixture (PR EOS)......................................................... 70
p_{cm}  Critical pressure of mixture (PR EOS).............................................................. 70
\omega_m  Acentric factor of mixture (PR EOS).............................................................. 70
\omega_i  Acentric of gas component i (PR EOS).......................................................... 71
p  Absolute pressure (PR EOS).................................................................................... 71
T  Absolute temperature (PR EOS).............................................................................. 71
V  Specific molar volume (PR EOS)........................................................................... 71
\mu_g  Gas viscosity (cp)............................................................................................. 71
\mu_g  Gas viscosity (cp)............................................................................................. 72
cp  Centipoises (cp)................................................................................................... 72
Y_{C_2H_6}  Mole fraction of C_2H_6............................................................................... 75
MW_{C_2H_6}  Molecular weight of C2H6 ................................................................. 75
1. INTRODUCTION

Industry has been able to economically develop the unconventional shale gas and tight gas reservoirs within United States and Canada in the past decade. The technologies of horizontal drilling with the continuous advancements in the multi-zone completion placing the multiple fractures along the wellbore have proven to be the significant keys to unlock those tight formations, which were previously uneconomic.

Among the horizontal completion methods applied, two distinct completion Methods: Plug-and-perf (P-n-P) and Open Hole Sleeve Multi-stage System (OHMS)-have been used effectively in almost every unconventional or tight gas play. Both methods have their own operational advantages and disadvantages, which the operators must be aware of before selecting one method over the other. OHMS is a relatively new and rapid completion system compared to the more mature P-n-P method. Nevertheless, OHMS has not yet gained a wide spread reception across the industry because of several factors (Casero, 2013).

Several authors have published papers comparing the P-n-P and OHMS completion methods. These publications compare a number of measures including initial production rate, ultimate recovery and long-term production rate. These publications do not agree in the same direction. Some authors concluded that one completion method is superior to another, while some concluded there is no significant difference in the term of production from both completion methods.

Augustine (2011) used a two-dimensional numerical model to compare production from both completion methods. Although, this model has many limitations, it has been cited in several subsequent publications as the only analytical method for comparing the two completion methods and answering this controversial issue.

This study aims to develop a three-dimensional Computational Fluid Dynamics (CFD) model to compare the steady-state natural gas production between P-n-P and OHMS completion methods in an unconventional reservoir. The parametric studies are performed and compared with the results from Augustine (2011).
1.1. UNCONVENTIONAL RESERVOIRS: TIGHT GAS AND SHALE GAS

In the United States, government regulation of coal has led to an increased demand for natural gas. This has led oil and gas operators to explore and develop more unconventional (lower permeability/source rock) type reservoirs using more advanced technologies.

Tight gas and shale gas are the two most common types of unconventional gas reservoirs that have been developed extensively in United States, Canada and other parts of the world. Although tight gas and shale gas may differ in terms of formation characteristics, reservoir analysis and production mechanisms, both resources require development with horizontal drilling and hydraulic fracturing in order to produce economically.

In order to develop a model of tight gas production through a horizontal well completion, it is first important to identify the reservoir characteristics and production mechanisms of a tight gas reservoir.

1.1.1. Tight Gas. A recognized definition of tight gas is “a reservoir that cannot produced at economic flow rates nor recover economic volumes of natural gas unless the well is stimulated by a large hydraulic fracture treatment or produced by use of horizontal wellbore or multilateral well bores.” Tight gas was also given the political definition by U.S. government in 1970’s as the gas reservoir, which has the permeability value less than 0.1 milliDarcy. This definition was created to determine which wells will receive federal or state tax credits. Both definitions are obscure and still cannot distinguish tight gas from shale gas or the others unconventional reservoirs, unless more reservoir characteristics are presented (Holditch, 2006)

Tight gas shares some reservoir characteristics with the conventional gas reservoirs. In both cases gas from another source rock migrated and was trapped in a sealed porous formation. Gas/ water contacts are usually absent but some wells produce water (Kennedy et al., 2012).

Holdich (2006) suggested that tight gas has no typical characteristics. It can be sandstone or limestone, deep or shallow, high or low pressure/temperature, blanket sands or lenticular, homogeneous or natural fracture and can contain a single or multiple layers.
Gas found in tight gas reservoir is a free gas, just as in conventional reservoirs, meaning it is stored either in the matrix/pore spaces of the rock or in natural fractures. Gas in pore spaces can be produced in the similar way as in the conventional gas reservoir. Tight gas reservoirs may produces dry gas while others may produce some wet gas. Due to the absence of a gas/water contact, gas is produced solely based on the solution drive mechanism.

**1.1.2. Shale Gas.** Shale gas refers to gas reservoirs which contains fine-grained, organic rich sedimentary rocks including shale but also composed of the other minerals like quartz and calcite (Kennedy et al., 2012). Passey et al. (2010) gives the definition of shale as extremely fine-grained particles, which are less than 4 micron in diameter but may contain variable amount of silt-sized particles (up to 62.5 micron). Some formations in the industry are categorized as shale because of their grain size, even though they contain a small amount of shale lithology.

Another character that defines shale gas is its gas permeability. Shale gas reservoirs typically fall within a permeability range from 1 to over 100 nanoDarcy or 0.000001 to over 0.0001 milliDarcy (King, 2012). In comparison to the permeability of conventional reservoir, which is approximately greater than 0.5 milliDarcy, gas in shale flows poorly. Hydraulic fracturing stimulation plays a significant role establishing the conductive paths, which enable the natural gas production in shale gas reservoirs. Figures 1.1 illustrates the permeability range of formations, which require hydraulic fracturing,
Shale gas is produced from organic-rich shale, which formerly was considered as a source rock not a reservoir rock. There is no trap similar to conventional gas reservoirs. The remaining hydrocarbon (gas) in shale was not expelled and was unable to migrate elsewhere. It does not contain a gas/water contact and unlikely produces water in a significant amount. It mostly produces dry gas (Kennedy et al., 2012).

Unlike tight gas, natural gas is stored in shale gas by three different ways (Kennedy et al., 2012):

1. **Free gas**: This is similar to tight gas. Gas is stored in the rock’s pore space/matrix and natural fracture.

2. **Sorbed gas**: Sorbed gas is divided as the adsorbed gas which is chemically bonded with the organic matter and mineral surface in natural fractures and the absorbed gas which is physically bonded with the organic matter and mineral surfaces in the rock’s matrix.

3. **Dissolved gas**: Gas is dissolved in a hydrocarbon liquid which is in kerogen.

In the early period of production, free gas is produced quickly leading to rapid production rate declines. Later, the sorbed gas, which has a greater volume than the free gas, desorbs from their attached surfaces, and the dissolved gas diffuses from the kerogen (Swami, 2012). Thus, the production modeling of shale gas reservoirs must take the consideration on these three different reservoir production mechanisms.
1.2. FRACTURE DIRECTION AND WELL ORIENTATION

Both fracture orientation and well orientation affect gas production in the hydraulic fractured wells. This chapter provides background information regarding subsurface stresses, their effect on fracture orientation, and the preferred fracture orientation in multi-stage fracturing horizontal wells.

1.2.1. Principal Stresses. All points in the subsurface are subjected to stress, due to the overburden of the rock and soil, and tectonic forces. These stresses are described with three orthogonal stress vectors, commonly referred to as the principal stresses. The three principal stresses include the vertical stress ($\sigma_v$) and the two horizontal stresses- the maximum horizontal stress ($\sigma_H$) and minimum horizontal stress($\sigma_h$). Figure 1.2 presents the three principal stresses.

Vertical stress is the stress due to the overburden formation, which in turn induces the horizontal stresses. Methods of determining the magnitude of the three principal stresses are presented in the literature (Economides et al., 1994).
In most areas of the world, the tectonic regime can be divided into three different regimes: Normal stresses regime, Strike slip regime, and Thrust regime. The vertical stress is highest in the Normal stress regime and will be the intermediate and lowest stress in the Strike slip and Thrust regime, respectively. Most reservoirs are in the Normal stress regime, which means their minimum stress is in the horizontal orientation (Bellarby, 2009).

1.2.2. Fracture Direction. Hydraulic fractures are created by breaking down the rock faces with greater force than the minimum rock stress. Hydraulic fractures propagate perpendicular to the least principal stress orientation. In the normal stress regime, the minimum stress is in the horizontal orientation, which means fractures will propagate in the vertical plane. In the region where the vertical stress is less than the horizontal stresses, the fracture will propagate horizontally. Figures 1.3 displays the fracturing propagation perpendicular to the minimum horizontal stress in the horizontal plane.

Figure 1.3. Fracturing propagation perpendicular to the minimum horizontal stress (Bachman et al., 2007)

1.2.3. Well Orientation. In Normal stress regime, hydraulic fractures in a vertical well (depth >>1500 feet) propagate vertically along the wellbore direction, which is perpendicular to the minimum horizontal stress (Figure 1.3). In a horizontal well drilled in a normal stress regime, the alignment of the horizontal wellbore relative to the
principal horizontal stresses determines fracture morphology as shown in Figure 1.4. A longitudinal fracture occurs when the horizontal well is drill in the direction of the maximum horizontal stress (Well A, Figure 1.4). However, if the well is drilled in the direction of the minimum horizontal stress, then a multiple, transverse fracture can be created as shown in Well B, Figure 1.4. Transverse fractures are initiated perpendicular to wellbore (Bahrami et al., 2013).

Figure 1.4. Longitudinal (left) and transverse fractures (right) in horizontal wells (Bahrami et al., 2013)

1.2.4. Well Performance from Different Types of Fractures.
Soliman et al. (1996) found similarity between well performance from longitudinal fractured horizontal well and fractured vertical well in tight gas formations. Soliman found two or more of transverse fractures are more effective to cover and drain the reservoir areas than the wells with longitudinal fractures. Hence wells with the transverse fractures produce gas more effectively than wells with longitudinal fractures in tight gas formations due to their ability to expose large reservoir area to drainage (Soliman et al., 1996).

Today, virtually all wells drilled in tight gas or shale gas reservoirs are oriented in the direction in the minimum stress (if known), so that transverse fractures may evolve from the stimulation treatment.
1.3. HORIZONTAL COMPLETION

In order to achieve the economic gas productivity from either tight gas or shale gas, large reservoir contact area is required. Horizontal drilling with a large numbers of transverse fractures intersect with the well has been accepted as the common approach.

Industry has developed numerous techniques to execute the multiple fracturing operations. Among these several techniques, which have been deployed, the two most common types of horizontal completion have been widely used: Plug-and-perf method (P-n-P) and Open Hole Multi-stage System (OHMS) completion method. These two techniques have a significant difference in terms of operations, speed, and well production (Snyder et al., 2011).

1.3.1. Plug-and-perf Method (P-n-P). This completion technique is the most common technique in the unconventional plays across the industry. P-n-P, is considered a mature approach. It has been used widely before OHMS. P-n-P is typically utilized in the cemented casing or liner in the horizontal section of the well. Figure 1.5 illustrates P-n-P method.

![Figure 1.5. Plug-and-perf method (Soliman et al., 2012)](image)

The fracture stimulation begins after the casing or liner was cemented in place. The operational sequence of P-n-P are as follows:
- The first set of perforations is created near the toe of the well by the coil-tubing-conveyed or the wireline-tractor-conveyed perforating gun.
- Acid is pumped before the treatment in order to clean up the debris after the perforation.
- Begin pumping the treatment to create the first fracturing stage.
- After the first stage is stimulated, the combination of select-fire-perforating guns and composite bridge plugs or frac plugs (wireline-deployed type) are pumped to place after the previous stage.
- The guns will fire after the bridge plugs/frac plugs are set.
- The pumping treatment for the current stage begins. Usually, each of pumping treatment stage will cover the group of perforation sets or “perforation clusters”.
- The processes of plug setting, perforation and pumping are repeated until all fracturing stages are completed.
- Mill out all composite plugs/frac plugs and let the well flows back to surface.

P-n-P provides the selective zone stimulations, which allow the fractures to be placed in the desired location.

In terms of production, gas from the reservoir flows to the wellbore through the perforation holes, which connect to the hydraulic fractures. The perforation holes generate pressure drop to the gas stream before reaching the wellbore. Moreover, the cased and cemented or off-fractured portions of P-n-P may block the gas production from the sandface, natural fractures and fissures which can contribute the gas flow.

1.3.2. Open Hole Multi-stage System (OHMS). OHMS was pioneered and commercially deployed in 2001 with the objective to improve the efficiency in terms of cost and time of fracturing treatment. It is utilized in competent and stable formations, where the wellbore stability is not a problem (Bachman, 2007). OHMS provides the ability to pump the fractured treatment, continuously with the technologies of ball activated sliding sleeves or “frac ports” and the size-specific actuation balls. While, the production liner is not cemented, the inter-frac zonal isolation can be obtained by the several type of casing annulus (open hole) packers such as mechanical or swellable.
packers. These technologies eliminate the need of bridge/frac plugs and perforations, which are used in P-n-P. OHMS can be performed in a single pumping operation, which saves cost, time and reduce HSE risk (Snyder et al., 2011). Figure 1.5 illustrates OHMS.

The operational approach of OHMS consists of the following steps:

- The OHMS assembly lower completion string is deployed into the well by drill pipe. The typical string consists of sequences of ball activated sliding sleeve and openhole packers, which have been spaced out by the production liner.
- The open hole packers are set in place without cementing the production liner.
- First frac port near the toe opens typically by the pressure activation. The pumping begins.
- The ball is dropped from surface and pumped through the string. Once the ball has landed on its designated sleeve’s seat, it will provide the isolation to the previous zone as well as shift the sleeve open to stimulate the current interval. The combination of balls and frac ports eliminate the need of bridge/frac plugs. The seats are designed to have a progressive larger internal diameter from toe to heel.
• At the end of the current treatment stage, the next ball is dropped to activate the next frac port.
• The processes are repeated in the continuous pumping treatment.
• After the last stage has been completed, the well can immediately flow back. All balls will flow back to the surface. In order to avoid the damage to the surface production equipment due to the fragment of balls, the coiled tubing milling is used. The seats are often milled out as well (Soliman et al., 2012).

OHMS lacks an ability to control the initiation point of fractures and its complex down-hole equipment requires high cost and operational understanding.

OHMS allows the gas from fractures flowing to the wellbore without any restrictions from the perforation holes like in P-n-P method. Since the off-fracture portions are not cemented in place, the off-fracture portions can contribute the gas flow to the wellbore.

Although the OHMS can be executed in one day compare to P-n-P, which requires a week to complete the operation, the operators still prefer to use P-n-P as their horizontal well completions. 70% of horizontal wells in Texas Panhandle tight gas use P-n-P as well as the majority of shale gas (Kennedy et. al, 2012).

1.4. FRACTURE PRODUCTIVITY

1.4.1. Concept of Productivity Index. The main purpose of productivity index is to be the parameter to quantify the well production from a single well and compare multiple wells performance as a function of drawdown.

In the steady state flow condition, a reservoir exhibits a constant pressure at outer boundary \( r_e \). The productivity is a ratio between the well production divides by the difference between static reservoir pressure and flowing bottomhole pressure. Equation 1-1 presents the productivity index for oil well (Wang et al., 2009).

\[
J_{oil} = \frac{q_o}{p_e - p_{wfb}}
\]
where \( J_{\text{oil}} \) is the productivity index in oil wells (STB/D-psi), \( q_o \) is oil flow rate (STB/D), \( p_e \) is the outer boundary pressure (psi) and \( p_{wf} \) is the flowing bottom hole pressure. In natural gas wells, the expression of productivity index is

\[
J = \frac{q}{p_e^2 - p_{wf}^2}
\]  

(1-2)

where \( J \) is the productivity index in gas wells (MSCF/D-psi\(^2\)), \( q \) is gas flow rate (MSCF/D) (Wang et al., 2009).

### 1.4.2. Notations of Hydraulic Fracture.

In this study, fracture can be characterized in term of length, width, height and conductivity. Fractured length is assumed to consist of two equal half-lengths \((x_f)\). This means the fracture grows equally perpendicular to the wellbore in the transverse fracture. This half-length is presented as the conductive length, where the gas flows through. It is not the hydraulic length, which is created physically by the hydraulic force.

The width of fracture is presented in the term of propped fracture width \((w)\). The propped fracture width is an average of fracture width and is simplified in this study to be a constant value along the fracture shape.

The propped fracture height \((h_f)\) is measured vertically in transverse fractures. Figure 1.6 illustrates the fractured half-length, propped fracture width and height in the horizontal well with a transverse fracture.
Fracture conductivity \((C_f)\) is a measurement how conductive of fracture is in the unit of md-ft. The fracture conductivity can be defined as (Wang et al., 2009):

\[
C_f = k_f w
\]  

(1-3)

where \(k_f\) is fracture permeability (md).

Another important parameter used in fracturing design is the dimensionless fracture conductivity \((C_{fd})\), which was introduced by Argawal et al. in 1979 and Cinco-Ley and Samaniego in 1981, and later used by researchers. The dimensionless fracture conductivity is:

\[
C_{fd} = \frac{k_f w}{k x_f}
\]  

(1-4)

where \(k\) is reservoir permeability (md).

**1.4.3. Improvement of Productivity Index.** The improvement of well productivity in the steady state condition can be expressed by the fold of increase (FOI). FOI is defined as (Economides et al., 1994):
\[ FOI = \frac{J}{J_o} \]  

Where \( J \) is the productivity index after the stimulation and \( J_o \) is the productivity index before the stimulation.

There are two methods to design the fracturing treatment in the steady state condition: Prats’ method and the McGuire-Sikora chart.

**1.4.3.1 Prats’ method.** Prats performed the analytical solution in 1961 (Economides et al., 1994). His solution is based on the steady state flow of incompressible fluid in the vertical well with the cylindrical drainage area. The propped fracture height is equal to the formation height.

Prats defined the term of relative capacity, \( a \):

\[ a = \frac{\pi k x_f}{2 k_f w} \]  

and the dimensionless effective wellbore radius (\( r_{wD} \)):

\[ r_{wD} = \frac{r'_w}{x_f} \]  

where \( r'_w \) is effective well radius, which is defined as

\[ r'_w = r_w e^{-s_f} \]  

where \( r_w \) is well radius and \( s_f \) is the equivalent skin due to the hydraulic fracturing.

The fold of increase from Prats can be defined as

\[ \frac{J}{J_o} = \frac{\ln(r_e / r'_w)}{\ln(r_e / r_w)} \]
Where \( r_e \) is the drainage radius or the outer boundary radius.

Figure 1.7 illustrates the curve from Prats’s study.

![Graph of Effective Well Radius vs. Relative Capacity Parameter](image)

**Figure 1.8.** Prat’s curve (Economides et al., 1994 retrieved from Prats (1961))

From Figure 1.7, Prats suggested that the hydraulic fracturing design should keep the value of relative capacity to be smaller than 1 because the dimensionless effective wellbore radius will decreases rapidly, if “a” is greater than 1. Another conclusion that he found is \( r'_{wD} \) approaches a maximum value of 0.5 as the relative capacity number ‘a’ decreases. In this area of Prats curve, fracture conductivity is very high relative to formation permeability, and is often referred to as an ‘infinite conductivity fracture’. In this area of Prats curve, only increasing of fracture length is beneficial; increasing fracture conductivity in the "infinite conductivity" portion of the curve cannot improve well performance.
1.4.3.2 McGuire-Sikora chart. Prats’ curve is used frequently today in hydraulic fracture design. However, Augustine (2011) refers to the hydraulic fracture reference curves developed by McGuire and Sikora in 1960, McGuire and Sikora (1960) studied the steady-state productivity from fractured vertical well in the square drainage area. The reservoir is assumed to be homogeneous with the fracture extends to the top and bottom of reservoir. Unlike Prats’ method, the compressible fluid was included in this study.

McGuire-Sikora Chart consists of the relationship between the relative conductivity:

\[ \text{Relative conductivity} = \frac{12w k_f}{k} \sqrt{\frac{40}{A}} \]  

(1-10)

Where \( A \) is the square drainage area (acres)

And the penetration ratio \( (I_x) \):

\[ I_x = \frac{x_f}{L_e} \]  

(1-11)

where \( L_e \) is the distance from well to the edge of square drainage area (ft).

The result fold of increase can be obtained from the function of

\[ \frac{J}{J_o} \left[ \frac{7.13}{\ln \left( \frac{0.472L_e}{r_w} \right)} \right] \]  

(1-12)

Figure 1.8 illustrates the McGuire-Sikora chart.
The McGuire-Sikora chart depicts different curves as a function of penetration ratio. However, each curve exhibits the same characteristic limit at very high fracture conductivity compared to the formation permeability (high relative conductivity). Increasing the fracture conductivity will not increase the fold of increase in producing rate. Only increasing the penetration ratio will increase the well’s production.

1.5. COMPUTATIONAL FLUID DYNAMICS IN WELL COMPLETION

Computational Fluid Dynamics (CFD) is the means of formally describing and analyzing fluid flow and other transport phenomena by the mean of computational based simulation. It is a powerful tool and has been used in wide range of industries. CFD’s prominence is its ability to perform the fluid flow analysis in the complicated flow geometries.

CFD typically requires high performance computing power to solve complex problems with large meshes. Physical models with large, complex meshes require powerful computers to solve the problem. In the past, CFD was not widely used because the hardware to perform CFD was very expensive. Rapid growth in computational capability has made CFD analysis more commonplace over the past decade. Moreover,
CFD has been developed progressively to provide high accuracy results, which are close to the reality. CFD produces a large number of results in very short time, thus CFD is suitable for the large amount of parametric studies and reduces considerable cost and time to do the laboratory experiments. Furthermore, CFD opens the opportunities to study the very large system, where the experiments are impossible to perform. These reasons have pushed CFD to be used widely in several industries since 1990s (Versteeg et al., 2009).

CFD has been used in oil and gas industry as an important design tool to examine flow in pipeline, surface equipment, drilling bit design and production string components such as inflow control valve (Byrne et al., 2011). However, CFD rarely has been used to study the near wellbore area where the complex fluid flow exists.

The conventional methods to design the fluid flow in well completion such as the analytical solutions and the nodal analysis are oversimplified. They lack of the ability to capture the complexity of well geometries in completion design at the near wellbore areas. Recently, several authors used CFD to perform the analysis on the near wellbore area. Jaminez and Chavez (2009) used CFD to examine the impact of perforation strategy and orientations of hydraulic fractures in the deviated well, in the heterogeneous oil reservoir. Byrne et al., 2011 used CFD to analyze the pressure lose due to the formation damage near wellbore due to the drilling mud. Sun et al.(2011) used CFD to study the skin factors in the perforation completion. Their study also included perforation-damaged zone and drilling damage. These CFD studies demonstrated how valuable CFD is in well completion design and this recognition led to the use of CFD in this study.

1.6. THESIS OBJECTIVE

Industry has referred to several publications regarding the selection of horizontal completions to maximize production in the unconventional plays, tight gas and shale gas. Augustine (2011) has been cited as the only analytical study focusing on the production comparison issue between the open-hole and cased-hole completion. However, the two-dimensional model in this publication has some limitations and provides some questionable results.
This study sought to create a three-dimensional, horizontal well production model incorporating a hydraulic fracture and comparing production through a P-n-P completion and the OHMS method of completion. The results are presented in the similar ways as Augustine’s and McGuire-Sikora’s works in order to compare the steady-state gas production from these two distinct completion methods. The parametric studies are performed on the propped fracture width ($w_f$), penetration ratio ($x_f/r_e$) and vertical to horizontal permeability ratio ($k_v/k_h$). The purposes of these parametric studies are to determine the effect of those parameters to the production comparison or “production penalty”. Finally, the results of this study are also compared with the results presented in Augustine’s publication (2011).
2. LITERATURE REVIEW

Several field studies have been published which compare production from P-n-P completions to OHMS completions in horizontal wells. This section reviews the historical work related to the two completion methods. The literature is divided into two categories – comparisons based on field studies and comparisons that utilize analytical methods.

2.1. FIELD OBSERVATION

The comparison of well performances between the wells completed with P-n-P and OHMS in unconventional plays have been studied and presented in several publications. The conclusions from these publications are not in the same direction.

Samuelson et al. (2008) studied the applications of P-n-P and OHMS in the Cleveland tight gas formation sand of Texas panhandle. The study utilized a statistical analysis of the production variables in Cleveland formation based on three months of continuous gas production. The cumulative production indicator called “best three months barrel of oil equivalent” showed wells with OHMS completions performed better than wells with P-n-P. The difference in production was greater than 25% in the areas of high reservoir quality.

Thomson et al. (2009) reviewed four different horizontal completion methods, which were used in tight gas reservoirs in northeast British Columbia, Canada. Among the four completion methods are P-n-P and OHMS. The production results from different completions were compared. Results of their work indicated that the different completion methods yielded similar production rates per interval compared, and any differences in production were negligible. The operators based their comparison on well construction and completion costs, and did not exclusively use production rate in the completion comparison.

Lohoefer et al. (2010) presented a long-term comparison of production between P-n-P and OHMS completions in Barnett shale. In this study, four wells were drilled in different areas of the Newark Field. Two wells were drilled in Tarrant County and two wells were drilled in Denton County. The two wells drilled in each area were twins,
drilled from the same pad and adjacent to one another. One well was completed P-n-P and the other well with OHMS. In the first set of wells in Denton County, the well completed with OHMS produced almost three and a half times more than the well completed with P-n-P during the three year period. In Denton County, the well completed with OHMS out produced the well with P-n-P by 57%, 37% and 51% after 12, 24 and 36 months, respectively.

Edwards, Braxton and Smith (2010) analyzed the gas production from the central area of Granite Wash, which is a tight sand reservoir in northern Texas Panhandle and western Oklahoma. In the central area, referred to as Northwest Mendota and Hemphill Fields, the average cumulative twelve month gas production per well from 24 wells completed with P-n-P, were compared with 6 wells completed with OHMS. The twelve month observation showed the wells completed with OHMS outperformed the wells completed with P-n-P by 33%.

Wilson et al. (2011) compared the daily gas production rates from 15 wells in two different geographical areas within the Lower Montney formation in a tight gas field southeast of Dawson Creek, British Columbia. The first area (Pod1) consisted of 4 wells competed with OHMS and 3 wells completed with P-n-P. After 14 months, the average daily gas production rate from each stage showed the wells completed with OHMS had an average daily production rate 38% better than wells completed with P-n-P. In the second area (Pod 2), there were 4 wells completed with OHMS and 4 wells completed with P-n-P. After 16 months of production, the average daily gas production rate of wells completed with OHMS was 28% greater than the daily gas production of wells completed with P-n-P. Although, OHMS outperformed P-n-P completions in both areas on average, not all individual OHMS completions outperformed individual P-n-P wells.

Snyder and Seale (2012) compared the average cumulative gas production from a group of wells completed in the Marcellus shale. The first group was in the southwest area of Marcellus shale in Washington County, Pennsylvania. Another group was in the northeast area of Marcellus shale in Susquehanna County, Pennsylvania. In each area, only a small numbers of wells were completed with OHMS, while the majority of completions were P-n-P. In southwest area, the wells completed with OHMS outperformed the well completed with P-n-P by 43%, 84% and 80% after 6, 12 and 24
months, respectively. In northeast area, the wells completed with OHMS completed with OHMS outperformed the wells completed with P-n-P by 3% and 37% after 6 and 12 months, respectively.

Kennedy et al. (2012) referred to field studies conducted by several operators. Those studies generally concluded that there is no difference in well’s initial production, using either completion method.

2.2. ANALYTICAL METHOD

Augustine (2011) analyzed the difference of well production between openhole and cemented completion in the two dimensional reservoir simulator.

Augustine’s two dimensional reservoir model utilized the concept of “equivalent length”, which combined the flow resistance from the radial flow component together with the reservoir length (L). The radial flow components are displayed in the half edge-drive reservoir in Figure 2.1 where the term of reservoir radius (r_e), reservoir pressure (P_e), reservoir height (h), well radius (w) and well pressure (P_w) are defined.

![Figure 2.1. Half of edge-drive reservoir model (Augustine, 2011)](image)
The edge-drive reservoir model has no flow boundaries on the top and bottom boundaries with the constant reservoir pressure ($P_e$) applied to at the right-end of the model. The derivation to obtain the equivalent length assumed the flow is under a steady-state condition, where non-Darcy effects are not included. Equation 2-1 and 2-2 represents the steady-state, Darcy's flow in the radial flow and linear flow components appears in the half of edge-drive reservoir model, respectively.

$$Q = \frac{0.00708k_h h(P_e - P_w)}{2B_o \mu \ln(r_e / r_w)} \quad (2-1)$$

$$Q = \frac{0.001127k_h A(P_e - P_w)}{B_o \mu L} \quad (2-2)$$

where

$$A = 2r_e h \quad (2-3)$$

$Q$ is the oil flow rate (bbl/day), $k_h$ is the horizontal permeability (md), $B_o$ is the oil formation volume factor (res. bbl/ bbl) and $\mu$ is the oil viscosity (cp)

The term equivalent length ($L_{eq}$) is defined as:

$$L_{eq} = \frac{2}{\pi} r_e \ln(r_e / r_w) \quad (2-4)$$

Substituting Equation 2-3 and combining $L_{eq}$ from Equation 2-4 into the term $L$ into Equation 2-2 results in :

$$Q = \frac{0.001127k_h (h2r_e)(P_e - P_w)}{B_o \mu(L + \frac{2}{\pi} r_e \ln(r_e / r_w))} \quad (2-5)$$

After performing a separate analysis on the effect of vertical to horizontal permeability ratio ($k_v/k_h$) to the equivalent length term, Augustine modified the equivalent length by multiplying the equivalent length with the negative 1/3 power of
The value 2 in the numerator of Equation 2-6 represents the full-round flow from the both halves of the model.

The value 2 in the numerator of Equation 2-6 represents the full-round flow from the both halves of the model.

Figure 2.2 depicts the top half of Figure 2.1, with the view rotated so that the wellbore is now oriented along the bottom of the figure. Wellbore length is now denoted as \( W_{\text{res}} \). A single transverse fracture has been added on the left, perpendicular to the wellbore. In the simulator, the fracture height equals to the reservoir height. The \( k_v/k_h \) of fracture is equal to 1. This model uses the concept of mirror images, i.e. there is a top and bottom section to the left of the wellbore in Figure 2.1 which is not shown, but which is added to the simulator to complete the entire volume surrounding the wellbore. No flow boundaries are maintained throughout the volume, consistent with the manner shown in Figure 2.2. A 6 inch well diameter was used throughout this study.
Figure 2.2. 1/4 of two-dimensional reservoir model (Augustine, 2011)

In Figure 2.2, $L_f$ refers to the fracture length, $L_{res}$ refers to the reservoir length, $W_f$ refers to the fracture width and $W_{res}$ refers to the reservoir width and the length of the wellbore. From these parameters, the penetration ratio, $L_f/L_{res}$, and reservoir aspect ratio, $W_{res}/L_{res}$ of the two dimensional model are calculated.

The results of the simulation were intended to be presented in a manner similar to the McGuire-Sikora curves. The parametric studies included the analysis on vertical to horizontal permeability ratio, reservoir height, reservoir aspect ratio and penetration ratio.

The parameters in this analysis consisted of reservoir permeability, ranging from 1 milliDarcy (md) to 1 nanoDarcy (nd), fracture permeability of 10, 100, 1000 Darcy, vertical to horizontal permeability ratio of 0.001, 0.01, 0.1 and 1.0, reservoir heights of 10, 100, 200 and 500 ft, an aspect ratio of 0.1, 0.25, 0.5 and 1.0 and the penetration ratio of 0.25, 0.5 and 0.75.

The results were shown with two curves depicting the performance of openhole and cemented completions. Each curve was plotted as a relationship between the “redefined relative conductivity” and the fold of increase ($J/J_0$). Figure 2.3 illustrates the
example of Augustine’s results from the case of $k_v/k_h=0.01$, reservoir height = 100 ft, aspect ratio = 0.50 and penetration ratio = 0.5.

Figure 2.3. Example of Augustine’s results (Augustine, 2011)

The redefined relative conductivity, as per Equation 2-7, was defined by Augustine with an intention to eliminate the propped fracture width from the classical dimensionless conductivity ($C_{fd}$).

\[
\text{Redefined relative conductivity} = \left( \frac{k_f w_f}{k_{res} L_f} \right)^{0.5} \quad (2-7)
\]

where $k_f$ is the fracture permeability (md), $k_{res}$ is the reservoir permeability(md), $w_f$ the is propped fracture width (in) and $L_f$ is fracture half-length (ft). Because of the redefined relative conductivity was used, the parametric study of fracture width was not presented, although it was mentioned in his publication.
According to Augustine’s conclusions, by assuming the equivalent fracture geometry and without the presence of natural fracture, it is physically impossible for a cemented completion to out-produced an openhole completion. These results show a significant difference of production for the P-n-P completion, which was defined as “production penalty” in the reservoir range of milliDarcy to microDarcy and the negligible production penalty in the range of microDarcy and nanoDarcy.
3. CONCEPT OF STUDY

3.1. RESERVOIR SELECTION

Although both P-n-P and OHMS completions have been used in tight gas and shale gas reservoirs, this study focuses on a comparison of the two completion methods in tight gas reservoirs. A tight gas reservoir was preferred because

- Production from tight gas can be modeled with Darcy’s flow equation whereas production from shale reservoirs requires a specific understanding of the fractures containing gas, extent of fractures present, and their connection to the completion. Gas production is a function of gas stored in the fractures and desorption of gas from the shale, and is a complex modeling problem which CFD may not be able to model correctly. This is discussed in recommended future work.

- The study by Augustine (2011) concluded that the greatest difference in production between P-n-P and OHMS occurs in the reservoir permeability range of milliDarcy to microDarcy, which is the permeability range for most tight gas reservoirs. The production penalty between completions methods lessens in the nanoDarcy permeability range of shale reservoirs (as $k_{res}^{-0.5}$ decreases, Augustine function on the x-axis of Figure 2.3 increases, showing less difference between the two types of completions). Hence, it was believed that CFD modeling should demonstrate a production difference for tight gas reservoirs.

- The algorithms in CFD are still not able to simulate the production mechanisms from sobbed gas and dissolved gas in shale, correctly. They required more understanding than the free gas flows in porous media, which is the current feature in CFD.

From these reasons, it was decided to compare the P-n-P and OHMS completions using a tight gas reservoir model. Details of the tight gas reservoir used in this study are presented in Chapter 5.

3.2. CONCEPTUAL MODEL

In this study, a horizontal well with a transverse fracture producing from a tight gas reservoir has been modeled using full 3D geometry with CFD software. The
horizontal well is modeled first without the fracture, and production is calibrated with standard flow equations as described in this section. A transverse fracture is added to the model, and then P-n-P completion is compared with OHMS. Only short portion (300 ft) of horizontal well is used in this study due to the limited computational resources.

3.2.1. Concept of Un-stimulated Model. The un-stimulated model was first developed using the “edge-drive reservoir” model of Augustine (2011). In this model, the horizontal well is placed in the rectangular drainage area of the homogeneous formation without gas-water contact. No flow boundaries are located at the top and bottom of the model. Free-gas in reservoir flows by the natural convection from the difference in the reservoir pressure ($p_e$) and wellbore pressure ($p_{wf}$) or a drawdown. The reservoir pressure applies from the outer boundary radius ($r_e$). The horizontal well is assumed to be the perfect cylindrical shape and located at the center of the reservoir model.

The terminologies of reservoir dimensions such as reservoir thickness ($h$) and horizontal well length ($L$) in this study are different from those defined in the work of Augustine (2011). They were changed to match with the terminologies defined by standard horizontal well flow equations (Eq. 5-1 and 5-4). Figure 3.1 illustrates the conceptual model of the un-stimulated horizontal well in this study.

Figure 3.1. Conceptual model of the un-stimulated well
3.2.2. **Concept of Stimulated Well.** The stimulated well is developed from the un-stimulated model. In this study, a single, constant-width transverse fracture is placed intersecting with the horizontal well at the middle of model (L/2). The propped fracture height (h_f) is assumed to be equal to the reservoir thickness (h). Fracture half-length (x_f) is less than the outer boundary radius (r_e). The connectivity between well and fracture depends on the completion type. The details of completion modeling within CFD are discussed in Chapter 6. Figure 3.2 illustrates the conceptual model of the stimulated well.

![Conceptual model of the un-stimulated well](image)

**Figure 3.2.** Conceptual model of the un-stimulated well

According to the new notation on the drainage radius (r_e) given in Figure 3.2, the penetration ratio (I_x) can be expressed as:

\[
I_x = \frac{x_f}{r_e}
\]  

(3-1)
4. CFD AND FLUENT

4.1. CFD OVERVIEW

Computational Fluid Dynamics (CFD) is the analytical technique using the numerical algorithms to solve the fluid flow problems with the aid from the computational software. CFD allows the users to perform the “numerical experiments” in the computer programs.

CFD is considered as the “third approach” of the fluid dynamic studies (Anderson, 1995). CFD has grown besides the pure theory and the pure experimental approaches and does not replace any of these classical approaches. CFD is most likely used to examine the problems in term of qualitative but can be qualitatively used, if the solutions are very accurate enough.

All CFD packages or codes contain the common three elements (Versteeg et al., 2009):

4.1.1. Pre-processor. Pre-processor is the input of the CFD package. Typically, it is designed to be the user-friendly interface and convert the input to the solver. The stages of pre-processes are:

- Define the geometry of interest. This geometry is called the “domain” in the CFD. Normally, the geometry can be created with a built-in computer-aided design (CAD) interface or an external modeler
- Sub-divide the main physical model into sub-domains by placing the grid or “mesh” on the domain. The result of this stage is to create the small cells or “control volumes”
- Specify the boundary conditions at the domain
- Define the appropriate fluid properties to the problem

Each cell contains a “node” on the inside. The node is where physical fluid properties such as the velocity, pressure and temperature of cell located. In general, the accuracy of the solution is a function of the number of cells, i.e. a greater number of cells yield a better solution. However, there is a high computing requirement associated with large meshes and number of cells and longer solution times are needed. CFD users must design the proper mesh on the domain to optimize between accuracy and cost.
CFD packages are typically equipped with the libraries of fluid/material properties, which accommodate users in the wide range of industries.

4.1.2. Solver. After the solver receives the input from the pre-processor, it will perform the following steps:

- Evaluate domain and transforms the governing equations into the set of partial differential equations
- Perform the discretization on the partial differential equation to obtain the algebraic equations
- Iteratively solve algebraic equations to obtain the final solution

Each CFD package has its own unique technique of domain evaluation and discretization.

4.1.3. Post-processor. The post processor is a visualization tool. With the high credible graphical hardware, the post-processor can visualize:

- Contour plot of flow properties
- Vector plot
- Surface plot
- Particle tracking
- Dynamic result displays

4.2. GOVERNING EQUATIONS IN CFD

4.2.1. Finite Control Volume. All problems in CFD are all involved with how to define the fluid flow in the finite control volume. The finite control volume is the closed surface with the fixed boundary or control surface (Anderson, 1995). The finite control volume is not required to be in an exact location (space). It can travel with the fluid elements. Instead of looking to the whole fluid flow system, CFD limits the fluid flow into the limit region, where the CFD applies the techniques to obtain the integral form of the fluid properties. The integral form can be discretized into the partial differential equations later. In the fixed-space finite control volume, which is the scope of this study, both integral and partial differential equations represent the conservation of fluid properties. Figure 4.1 illustrates the fixed-spaced finite control volume with the fluid flows.
4.2.2. **Continuity Equation.** In the finite control volume or control volume, the fluid flowing through all control surface must follow the mass conservation law (Versteeg et al., 2009):

\[
\text{Net rate of mass flow out} = \text{Rate of decrease of mass inside} \quad (4-1)
\]

Figure 4.2 illustrates the mass flow in and out of the control volume.
The final form of the mass conservation law, the mass conservation equation, is represented by Equation 4-2 (Versteeg et al., 2009):

$$\frac{\partial \rho}{\partial t} + \left[ \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} \right] = S_m$$  \hspace{1cm} (4-2)

or in the gradient form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = S_m$$  \hspace{1cm} (4-3)

where $\rho$ is the density of fluid, $t$ is time,
$\delta x, \delta y$ and $\delta z$ are the distances in x, y and z direction, respectively,
u, v and w are the fluid velocities in the x, y and z direction, respectively,
and $S_m$ is mass added to the continuation phase.

The derivation of mass conservation equation can be found from several CFD textbooks such as Anderson (1995) and Versteeg et al. (2009). In the CFD packages, the mass conservation equation is called “continuity equation”.

### 4.2.3. Momentum Equations.

Besides the mass conservation law, the fluid must also follow the second law of Newton. For fluid flow, Newton’s second law can be stated as (Versteeg et al., 2009):

$$\text{Rate of increase of momentum fluid particle} = \text{Sum of force on fluid particle}$$  \hspace{1cm} (4-4)

The rate of increase momentum per unit mass in x, y and z direction are given as (Versteeg et al., 2009):

$$\rho \frac{Du}{Dt}, \rho \frac{Dv}{Dt}, \rho \frac{Dw}{Dt}$$  \hspace{1cm} (4-5)

where term $Du/Dt, Dv/Dt$ and $Dw/Dt$ define rate of change of velocity per unit mass in x, y and z direction, perspective.
Forces on the fluid are the results from the product of stress and area. The stress consists of pressure \((p)\), a normal stress, and the viscous stress term \((\tau_{ij})\). The subscripts \(i, j\) represent the stress components act in the \(j\) direction on the surface normal to the \(i\)-direction. Figure 4.3 illustrates the stress components in \(x\), \(y\) and \(z\) planes.

![Figure 4.3. Stress components (Versteeg et al., 2009)](image)

The momentum equations can be derived by the principle from Equation 4-4. In the \(x\) direction, the momentum equation for the \(x\)-component is (Versteeg et al., 2009):

\[
\rho \frac{Du}{Dt} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \tag{4-6}
\]

In the \(y\) direction, the momentum equation for the \(y\)-component is (Versteeg et al., 2009):

\[
\rho \frac{Dv}{Dt} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \tag{4-7}
\]

In the \(z\) direction, the momentum equation for the \(z\)-component is:
\[
\frac{\rho D_{w}}{D_{t}} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial (-p + \tau_{zz})}{\partial z} + S_{M_{z}}
\]

(4-8)

where \(S_{M_{x}}, S_{M_{y}} and S_{M_{z}}\) are the body force in \(x, y\) and \(z\) direction, respectively.

**4.2.4. Navier-Stokes Equations.** According to the momentum equations in section 4.2.3, the unknown terms of \(\tau_{ij}\) are not feasible. They need to be defined by the suitable fluid model. In this study, which the isotropic Newtonian fluids are used, the viscous stresses are proportional to the rate of deformation in fluid element. Refer to the derivations by Versteeg et al.(2009), Equation 4-6 can be rewritten as:

\[
\frac{\rho D_{u}}{D_{t}} = -\frac{\partial p}{\partial x} + \text{div}(\mu \nabla u) + S_{M_{x}}
\]

(4-9)

where \(\mu\) is the dynamic viscosity.

The term \(S_{M_{x}}\) in Equation 4-9 is added with the contribution from the viscous stress in order to simplify the equation. Furthermore, Equation 4-7 and 4-8 can be derived in the similar manner (Versteeg et al., 2009):

\[
\frac{\rho D_{v}}{D_{t}} = -\frac{\partial p}{\partial y} + \text{div}(\mu \nabla v) + S_{M_{y}}
\]

(4-10)

\[
\frac{\rho D_{w}}{D_{t}} = -\frac{\partial p}{\partial z} + \text{div}(\mu \nabla w) + S_{M_{z}}
\]

(4-11)

Equation 4-9, 4-10 and 4-11 are in the useful form of Navier-Stokes equations. They are used to control the momentum behavior of the Newtonian fluids in CFD.

**4.2.5. Porous Media Terms in Momentum Equations.** In this study, the majority of the domain (the wellbore model) is defined as one or more regions of porous media. Thus, this section briefly discussed how CFD accounts for the properties of porous media in the momentum equations.
When porous media properties are assigned in the pre-processor, CFD assigns the additional force \( S_i \) into the body force term of momentum equations, where the subscript \( i \) refers to the direction \( i \). As a result, the fluids are treated like the bulk fluids with more resistance.

In simple homogeneous porous media with fluid flow in the \( i \) direction, the additional force is defined by Equation 4-12. The Darcy’s law is the first term on the right side of Equation 4-12 and the inertia loss term is the second term on the right side of equation 4-12 (ANSYS FLUENT User’s Guide, 2011):

\[
S_i = -\left( \frac{\mu}{\alpha} v_i + C_i \frac{1}{2} \rho |v| v_i \right)
\]  

(4-12)

where \( \alpha \) is the permeability, \( v_i \) is the superficial velocity in the \( i \) direction, \( C_i \) is the initial resistance factor in the \( i \) direction, \( |v| \) is the magnitude of velocity and \( v_i \) is the vector of velocity in the \( i \) direction.

If the assumption of laminar flow is used, term \( C_i \) in Equation 4-12 is considered to be zero. The remaining equation is the Darcy’s law, which can be written in term of pressure drop as (ANSYS FLUENT User’s Guide, 2011):

\[
\nabla p = -\frac{\mu}{\alpha} \nabla v
\]  

(4-13)

The gradient form in Equation 4-13 can be expressed in three coordinate directions \((x, y, z)\) as per Equation 4-14, 4-15 and 4-16. CFD calculates the pressure drops through the porous media based on these equations (ANSYS FLUENT User’s Guide, 2011)

\[
\Delta p_x = \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x 
\]  

(4-14)

\[
\Delta p_y = \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y 
\]  

(4-15)
\[ \Delta p_z = \sum_{j=1}^{3} \frac{\mu}{\alpha_{ij}} v_j \Delta n_z \]  \hspace{1cm} (4-16)

where \( \Delta n_x, \Delta n_y \) and \( \Delta n_z \) are the thickness of porous media in \( x, y \) and \( z \) direction, respectively. The subscript \( x, y \) and \( z \) indicate the flow direction in \( x, y \) and \( z \) axis, respectively.

**4.2.6. Equation Of States.** In CFD, the unknown variables of pressure (\( p \)), density (\( \rho \)), temperature (\( T \)) are linked by the assumption of thermodynamic equilibrium. Under this assumption, the properties of fluids are simultaneously changed, corresponding to their current conditions. The simple equation of state is (Versteeg et al., 2009):

\[ p = p(\rho, T) \]  \hspace{1cm} (4-17)

The detail of selected equation of state shall be discussed again in section 5.2

**4.2.7. Energy Equation.** Although, the heat transfer problem is not included in this study, the definition of energy equation shall be given here.

The energy equation in CFD is derived from the first law of thermodynamic, which is (Versteeg et al., 2009):

\[
\text{Rate of increase in fluid} = \text{Net rate of heat added to fluid} + \text{Net rate of work done on fluid}
\]  \hspace{1cm} (4-18)

The derivation of the energy equation can be found in the CFD textbooks mentioned in section 4.2.2.
4.3. **DISCRETIZATION SCHEME: FINITE VOLUME METHOD**

Discretization is the process of approximation on the differential or integral equations, which contain the variables that are considered as the continuous values throughout the domains. The discretization expresses these continuum phases by defining the values at the specific finite points in the form of numerical solutions. In CFD packages, there are several types of discretization such as

- Finite difference method
- Finite element method
- Finite volume method

The majority of the well-known CFD packages, including “FLUENT” which is used in this study, utilize the finite volume method as their discretization scheme.

Finite volume method was the latest discretization scheme compares to the finite different and finite element method (Bakker, 2002-2006). Its advantages include

- Always yield mass, momentum and energy conservation even though the discontinuity occurs in the domain.
- Is not limited by cell shape and allows coarse grids to be used in the domain. This is not possible in the finite different method.
- Calculate the turbulence flow, high speed flow with less memory compared to the finite element method.

The finite volume method divides the problem domain into the finite number of small cells or control volumes by placing grids or meshes over the domain. Grids define the finite boundary of each control volume where the computational node located at the center. Each computational node contains the properties of the control volume. Figure 4.4 displays the sub-divided domain with the computational nodes.
The basic steps of the finite volume method are:

- Divide domain into the small non-overlapped control volumes
- Integrate the differential equations over the control volume to find the net flux over the control volumes faces
- Apply the integral forms with the divergence theory. This process requires the values of flux at the control volume faces, which is obtained by the interpolation or approximation
- The previous step results in the set of linear algebraic equations. Each control volume provides a unique set of algebraic equations.
- Obtain results by solving the equations iteratively or simultaneously.

The in-depth theories behind the finite volume method can be found in the textbooks from Verteeg et al. (2009).
4.4. CFD SOLVER

4.4.1. Types of Solver. In CFD, two solver algorithms are available (ANSYS FLUENT Theory Guide., 2011). These include the pressure-based solver and the density-based solver.

4.4.1.1 Pressure-based solver. The pressure-based solver was traditionally invented for low speed problems with the incompressible fluids but recently can be used beyond its original intended scope. In the pressure-based solver, the velocity field is obtained by solving the pressure correction equations under the mass conservation law. The pressure correction equations are derived from the continuity equations and momentum equations together. The repeated iterations are performed in these processes because the governing equations are non-linear and coupled together. Lastly, the iteration stopped after the solution reaches convergence. The common coupling methods of the continuity and momentum equation in the steady state condition such as SIMPLE, SIMPLER and SIMPLEC are available in CFD packages.

4.4.1.2 Density-based solver. Density based solver was invented to solve high speed flow of incompressible fluid. The difference between the density-based solver and the pressure-based solve is the density-based solver can solve all variables simultaneously without the multiple steps required by the pressure-based solver.

4.4.2. Steady-state Iterative Scheme. The steady-state iteration in CFD is controlled by the under-relaxation factor(\(\alpha\)). The change of current solution variable (\(\phi\)) from the old solution variable (\(\phi_{old}\)) is defined by (ANSYS FLUENT Theory Guide, 2011):

\[
\phi = \phi_{old} + \alpha \Delta \phi
\]

where

\[
\Delta \phi = \phi - \phi_{old}
\]

The proper setting in the under-relaxation factor will reduce \(\Delta \phi\).

4.4.3. Residual Sum. In the pressure-based solver, CFD records the convergence history in terms of residual sum (\(R^\phi\)). The residual sum will go to zero in an infinite precision computer as the solution converges. In a normal computer, the residual
sum reduces to a very small value such as $8(10)^8$ or $5(10)^{-12}$ when a solution is reached. The actual residual number depends on the problem solved, but will generally be a number raised to a large negative exponent.

After discretization, the solution variable ($\phi$) in cell P: $\phi_p$ can be expressed as (ANSYS FLUENT Theory Guide, 2011):

$$a_p \phi_p = \sum_{nb} a_{nb} \phi_{nb} + b$$  \hspace{1cm} (4-21)

where $a_p$ is the central coefficient, $a_{nb}$ is the influence from the neighboring cells and $b$ is the contribution of the constant part of the source term.

The residual sum ($R^\phi$) can be defined as (ANSYS FLUENT Theory Guide, 2011):

$$R^\phi = \left| \sum_{CellP} \sum_{nb} a_{nb} \phi_{nb} + b - a_p \phi_p \right|$$  \hspace{1cm} (4-22)

Although, the default convergence criterion in CFD is a low magnitude of residual sum, CFD users should not use the residual sum alone to judge the convergence. Changes in the output parameters must also be monitored in combination with the residual sum to ensure the simulation has reached true convergence.

4.5. ANSYS-WORKBENCH AND FLUENT

4.5.1. Overview. ANSYS-workbench is a project-based software which employs schematic representations of the workflow. It is a visual connection between pre-processor, solver and post-processor within the analysis system. The analysis systems are presented in the flowchart-like diagrams is shown in Figure 4.5.
ANSYS-workbench was created with the intention to provide integrity of the simulation modules. In the past, the pre-processor, solver, and post-processor operated separately in the different software. Previously, e conversions of data were required and users needed to execute this manually. ANSYS-workbench eliminates the converting processes into the robust workflow, which is easier to use and manage. The current application of workbench also provides the multi-physics coupling such as the combination of dynamics fluid flow solver or CFD with the mechanical structure solver.

In ANSYS workbench, several CFD solvers are available. This study utilized the workflow of the CFD solver “FLUENT”. CFD has been used intensively in petroleum industry as the primary CFD solver for many applications as documented in the literatures (Byrne et al., 2011 and Sun et al., 2011).

**4.5.2. Components in FLUENT.** FLUENT is available in ANSYS workbench as a complete analysis system or as an individual solver which needs to receive the input from the other pre-processors by creating the flowchart links, manually. This section describes all components in the FLUENT analysis system. Figure 4.6 illustrates the analysis system of FLUENT in ANSYS workbench.
FLUENT analysis system consists of:

**4.5.2.1 DesignModeler.** DesignModeler is the geometry builder, which can be accessed through the “Geometry” module of the analysis system. This is the first pre-processor, where the problem geometry or domain is defined. The models in DesignModeler can be built in 2D or 3D format, depending on the user’s choices. In addition, DesignModeler allows users to build either highly complex geometries with intricate details, or model a very large domain. Figure 4.7 illustrates the DesignModeler module.
Figure 4.7. DesignModeler module

4.5.2.2 Meshing  Meshing is the second pre-processor, where the geometry from the DesignModeler is imported and divided to the discrete cells or control volumes as discussed in section 4.3. Meshing can be accessed through the “Mesh” module. Figure 4.8 displays the Meshing module.

Figure 4.8. Meshing module
The grid or mesh can be controlled and applied either globally (to the entire domain) or locally, over select portions of the domain. In 3-dimensional models, the shapes of discrete cells are varied depending on the meshing approach used. Four types of cells are available in meshing (ANSYS Lecture 3, 2012):

- Tetrahedral
- Pyramidal
- Prismatic
- Hexahedral

Figure 4.9 illustrates 4 types of 3 dimensional meshing cells.

![Meshing cells](image)

Figure 4.9. Meshing cells (ANSYS Lecture 3, 2012)

The mesh qualities must be verified before the sub-divided domain is exported into the next pre-processor. The mesh qualities such as the mesh density, aspect ratio, skewness and orthogonal quality have their own ranges of quality, which users must be aware of. Users can evaluate the statistical data of mesh qualities inside Meshing. The recommended range of each mesh quality criteria can be found in the ANSYS Meshing user guide. Mesh qualities have the significant effects on the solution convergence and
solution qualities. Bad physical problem descriptions and poor quality meshes can create difficulty in achieving solution convergence.

4.5.2.3 Setup & Solution. Setup is the last pre-processor located within FLUENT. Setup is where the boundary/zone conditions, material properties, flow regime, solver algorithm and iterative controls are defined. Figure 4.10 illustrates Setup module.

![Figure 4.10. Setup module in FLUENT](image)

Boundary describes the surfaces of the domain. The mandatory boundary conditions are the inlets and the outlets of the domain. Other types of boundaries are available, such as walls, where fluid cannot flow through, or symmetry planes, where the domains can be modeled by splitting the domain into symmetrical halves or quarters, to allow a smaller portion of the main problem to be solved with less cost and time.

Zone conditions are the properties of the fluid zones. Several zones can be defined in the domain’s body because the domain is not homogeneous in some studies. The available function of porous media and source terms can be assigned in to the specific zones.
The material properties are readily available in the pre-built libraries within FLUENT. The prebuilt libraries consist of the common fluid/solid properties in the industries. Moreover, the users are allowed to create their own material libraries with their interest materials.

Flow regimes of laminar flow and turbulence flow with the different turbulence models are available within FLUENT. A turbulence model must be selected with caution to match with the problem.

Solver algorithms setup provides the choices of pressure/density-based solver, steady/transient flow, single phase/multi-phases flow, discretization scheme and iteration scheme. Different problems need to employ suitable solver algorithms in order to reach convergence.

After all setups are complete, the users can perform the simulation processes within this module. Before the iteration begins, users select the initializing methods. Good initialization results in reduced iterative time. After initializing, the user can start the iterations. The iterative values of governing equation are controlled by the under-relaxation factors, which can be adjusted by users. The iterations will stop depending on the convergence criteria desired by users. In the default setup, FLUENT stops the iterations when the residual sum of governing equations dropped down below three magnitudes ($10^{-3}$). Nevertheless, only residual sum alone does not present the true convergence, where the governing equations are fully conserved. Other parameters must be observed and judged together with the residual sum for the true convergence. Sometimes, the solution divergence does occur because of several reasons such as the nature of geometry, low meshing qualities and improper algorithms. So, the users must understand the nature of FLUENT solver as well as the simulation models in order to obtain the desired solutions.

The solutions are stored inside the software memory when the iterations have completed, the users can see the previous simulation results within the Solution module.

After all iterations have stopped, the post-processor in FLUENT is available for the users to review the overview properties, such as contours, streamlines, statistical properties at the specific boundaries and an animation of the simulations. Although, the
FLUENT post-processor is convenient and provides basic operations, some capabilities of this post-processor are still not versatile enough compare to the CFD-Post module.

4.5.2.4 CFD-Post. CFD-Post is the well-developed post-processor located as the last module of FLUENT analysis system in ANSYS-workbench. In Figure 4.6, CFD-post can be accessed through the Result module. Figure 4.11 displays CFD-Post with a graphical contour.

![Figure 4.11. CFD-Post](image)

CFD-Post works by importing the recent simulation data from FLUENT and providing the graphic user interphase (GUI) to the users. Most of applications in CFD-Post start with users specify their interest locations, which can be boundaries or fluid zones, then create the numerical expressions of their interest. Then, CFD-Post will generate the quantitative data of these interest locations into the reports or graphical renders as per users ‘assignments. CFD-Post also can display the graphical objects such as the vector plot, streamline, contours and volume rendering. CFD-Post is better than the FLUENT post-processor in term of report preparation and multi-files comparison.

4.5.3. Workflow in FLUENT. Figure 4.12 summarizes the ANYSIS FLUENT workflow discussed in the previous sections.
The workflow starts with the geometry modeling inside the DesignModeler. Then, users perform the mesh generation inside the Meshing. Users justify the mesh qualities by the statistical tools inside the Meshing and perform re-meshing, if needed, to obtain the satisfied meshing qualities. Users then move to the FLUENT set up and assign the simulation properties. The simulation can perform and should reach the convergence with proper setup. If the solution does not reach the convergence or provides unrealistic results after users performed the evaluation processes, then users must verify the sources of problems. Problems can be from the model, meshing or every single setup within FLUENT. These processes take considerable time to perform in order to obtain the final solutions. Lastly, users present their simulation data by the assist from the FLUENT post-processor or CFD-Post.

Figure 4.12. Flowchart of FLUENT analysis system
5. HORIZONTAL WELL MODEL VALIDATION

This section introduces the fundamental horizontal well production equations derived by Joshi (1988) to represent the horizontal well deliverability. These equations have been used to validate CFD models in this study. The PVT correlations for natural gas properties in petroleum engineering are also presented here along with the equation of states, which CFD utilizes to calculate the natural gas density.

5.1. HORIZONTAL WELL PRODUCTION

5.1.1. Horizontal Well Equation for the Incompressible Fluid. The horizontal well equation for the single phase flow of incompressible liquid, was introduced by Joshi (1988) and developed later by Economides et al. (1990). This equation mixes steady state flow in the horizontal plane and pseudo-steady state in vertical plane together without any formation damage, turbulence effects and stimulation. It can be defined as (Economides et al., 1994):

\[
q = \frac{k_h h (p_e - p_{wf})}{141.2 B \mu (\ln \left[ \frac{a + \sqrt{a^2 - (L/2)^2}}{L/2} \right] + \left( I_{ani} h \right) \ln \left[ \frac{I_{ani} h}{r_v (I_{ani} + 1)} \right])}
\]

(5-1)

where \( q \) is the surface volume flow rate (STB/d), \( k_h \) is the horizontal absolute permeability (md), \( h \) is the reservoir thickness (ft), \( B \) is the formation volume factor of liquid (res bbl/STB), \( L \) is the horizontal well length (ft), \( I_{ani} \) is defined as the index of anisotropy,

\[
I_{ani} = \frac{k_h}{k_v}
\]

(5-2)

where \( k_v \) is the vertical absolute permeability (md). The parameter ‘a’ is related to the drainage ellipse as,
\[
a = \frac{L}{2} \left\{ 0.5 + \left[ 0.25 + \left( \frac{r_{eh}}{L/2} \right)^4 \right]^{0.5} \right\}^{0.5} \quad \text{for} \quad \frac{L}{2} < 0.9 r_{eh} \\
\]

where \( r_{eh} \) is the distance to the outer boundary (ft)

### 5.1.2. Horizontal Well Equation in Gas Reservoir.
In an isothermal, single phase (dry gas) reservoir, Equation 5-1 can be written to describe the gas production from the horizontal well (Economides et al., 1994):

\[
q_{gas} = \frac{k_{\mu} h (p_e^2 - p_{wf}^2)}{1424 \mu \bar{Z} T \ln \left\{ \frac{a + \sqrt{a^2 - (L/2)^2}}{L/2} \right\} + \left( \frac{I_{ani} h}{L} \right) \ln \left( \frac{I_{ani} h}{r_w (I_{ani} + 1)} \right)}
\]

(5-4)

Where \( q_{gas} \) is the natural gas flow rate at surface (MCF/d), \( \bar{\mu} \) is the average gas viscosity between outer boundary and wellbore (cp), \( \bar{Z} \) is the average gas Z-factor between outer boundary and wellbore and T is the reservoir temperature(°F). All the gas properties are calculated using the correlations in section 5.3.2.1.

In high rate gas wells, the turbulence effect can be added into Equation 5-4. Since the turbulence effects are neglected in this study, turbulence is not included here.

### 5.2. BASIC MODEL DATA

The un-stimulated horizontal well model (section 3.2.1) in FLUENT must be validated before any completion models can be included. This section provides the basic information about the initial model validation and subsequent modifications.

#### 5.2.1. Reservoir and Natural Gas Data.
Among the available tight gas reservoir data available in the literature, this study uses data from Appleby North field in East Texas, taken from the publication by Magalhaes et al. (2007). The vertical to horizontal permeability is assumed to be 0.1 based on the permeability data provided. Natural gas compositions are required in all gas properties correlations as well as the equation of state inside FLUENT. Unfortunately, no gas compositions were available in the publication of Magalhaes et al. (2007). Thus, the author used gas composition from a
Fayetteville Shale well, deemed to be similar. Table 5.1 provides the reservoir data and natural gas compositions.

Table 5.1. Reservoir and natural gas data

<table>
<thead>
<tr>
<th>Reservoir and fluid properties</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reservoir rock</td>
<td>Sandstone</td>
<td>-</td>
</tr>
<tr>
<td>Net pay; h</td>
<td>60</td>
<td>ft</td>
</tr>
<tr>
<td>Horizontal absolute permeability; $k_H$</td>
<td>0.01</td>
<td>md</td>
</tr>
<tr>
<td>Vertical absolute permeability; $k_V$</td>
<td>0.001</td>
<td>md</td>
</tr>
<tr>
<td>Reservoir temperature; $T$</td>
<td>254</td>
<td>°F</td>
</tr>
<tr>
<td>Reservoir/Boundary pressure; $p_e$</td>
<td>2800</td>
<td>psi</td>
</tr>
<tr>
<td>Gas compositions</td>
<td>95% Methane, 5% Ethane</td>
<td>-</td>
</tr>
</tbody>
</table>

### 5.2.2. First FLUENT Horizontal Well Model.

The first horizontal well model was constructed based on the conceptual model in section 3.2.1. Because of the limited computational resources, the size of the model had to be reduced by splitting the domain into two mirror-image sub-domains, using a symmetry plane through the wellbore. This allowed the model to be much smaller, to fit the computation capabilities available in the research. After the model geometry was defined, the outer boundary radius ($r_e$) and horizontal well length ($L$) of 300 ft are selected because the combination of these two parameters results the valid solution from Equation 6-1. The concept of symmetry plane was applied a second time to reduce the size of model in half. Thus, the well length was reduced to 150 ft in the model. By placing the symmetry planes opposite to each other, the model should present the continuous series of horizontal well portion. Figure 5.1 depicts the symmetry planes used in the model.
The inlet is assumed to be the outer boundaries, which are applied by the constant reservoir pressure; $p_e$. The outlet is assumed to be the boundary of the horizontal well internal diameter. A 6-inch diameter horizontal well with a perfect cylindrical shape is assumed, and the well is fully openhole without any formation damage. A constant well pressure is assumed acting normally along the well boundary as shown in Figure 5.2.

The top and bottom boundaries of the model assumed to be the no-flow boundaries. The non-frictional wall boundaries are assigned to these boundaries in FLUENT. Figure 5.3 illustrates the overview of the horizontal well model in DesignModeler.
5.3. MODEL VALIDATION

The model validation phase is divided into two main parts:

- Incompressible fluid model
- Compressible fluid model

The incompressible fluid model is matched first because it verifies only the assumption of drainage geometry in comparison with the horizontal well equation (Eq. 5-1), and excludes the effect of gas compressibility which require specification of Z-factor and equation of state. Once the model is validated against the incompressible flow equation, the compressible model of gas flow is then matched with the assumption of horizontal well drainage given by Joshi (1988), verifying that flow results agree with this theoretical equation. In the validation, the fluid viscosity values used in FLUENT and the horizontal well equation are the same constant value.
5.3.1. Incompressible Fluid Model Validation. The first incompressible fluid model was constructed based on the geometry used by Augustine (2011). It was believed that if flow results obtained from FLUENT for this model matched results from fundamental flow equations using incompressible flow, then the model would be considered valid. If a match could not be obtained, then the flow model of Augustine would need to be changed. Validation solely determines the correct drainage pattern of FLUENT horizontal well model with the fundamental horizontal well equation (Eq.5-1).

5.3.1.1 Validation steps in the incompressible fluid model. This validation uses water to validate the drainage geometry of FLUENT model with the horizontal well equation (Eq. 5-1). The validation steps included:

- Perform meshes generation in Meshing. The meshes’ qualities are checked to ensure they are in the acceptable ranges. The horizontal well model after meshing process is displayed in Figure 5.4.

![Figure 5.4. Horizontal well model after Meshing](image)

- In FLUENT, the domain is a single zone with the porous media feature. The porous media, or porous zone, is assigned in FLUENT. The porous zone can be assigned in the cell zone conditions panel. When the porous zone check box is
checked and the viscous resistance is assigned, the zone is defined as a porous media in FLUENT. Figure 5.5 illustrates the cell zone condition panel.

![Cell zone conditions panel](image)

Figure 5.5. Cell zone conditions panel

- FLUENT’s viscous resistance is the inverse of permeability. The conversion obtaining viscous resistance (1/m²) from the permeability (k) in md unit is (Wang et al., 2009):

\[
\text{Viscous resistance (1/m}^2\text{)} = \frac{1}{k \text{ (md)} \times 9.9 \times 10^{-16}}
\]  

(5-5)

For the drainage area matching, values of gas permeability data given in chapter 3 are used as the water permeability. Direction 1, 2 and 3 are set as X, Y and Z.
direction in FLUENT, respectively. The model has two equal horizontal permeability values in x and y direction and the individual z permeability, which is in the vertical direction. Any non-Darcy effects are excluded.

- A perfectly cylindrical, horizontal well is assumed with no skin damage.
- The pressure-based, single phase, laminar flow and steady-state solvers are used. No gravitational force included. The energy equation is not activated, so no heat transfer include.
- The simulation assigns the different well pressure cases from 200, 500, 800, 1000, 1200, 1600, 1800, 2000, 2200 and 2500 psi.
- The water density of 998.2 kg/m$^3$ and viscosity of 1.003 cp are used. These properties are assumed to be constant pressure-independent values.
- The simulation can proceed following the procedures mentioned in section 4.5.
- In each well pressure case, the convergence criteria are defined by mass conservation and the non-reversible flow occurs. The mass conservation is displayed by creating the monitoring surface of the sum between all outlets (negative flows) and inlets (positive flows). The mass conservation, which dropped down than three magnitudes ($10^{-3}$) from the mass flow rate at the outlet, indicates the mass conservation and true convergence. Figure 5.9 illustrates the monitoring of mass conservation and outlet mass flow rate in FLUENT (kg/s) from $p_{wf} = 1000$ psi case. In Figure 5.6, the final mass conservation is significantly lower the mass flow rate at outlet. No indications of reversible flow appear. Furthermore, the residual sums of governing equations are gradually reducing. Most importantly, the mass flow rate at the outlet is now becoming a constant value. These are the signs on true convergence and the iteration can be stopped here.
The volume flow results, obtained from FLUENT, are compared to the volume flow rate obtained from Equation (5-1) in the unit of reservoir barrels per day (RVB/d) in the inflow performance relationship (IPR) format. Equation 5-1 is used without the formation volume factor of liquid (B) to obtain the unit of RVB/d.

Any errors that occur required a change in the drainage geometry.

The final drainage geometry model shall be analyzed later in the compressible fluid model.

The details of solver setup with the captions are all in Appendix B.
5.3.1.2 Inflow performance relationship (IPR). The definition of IPR is mentioned in this section because its principle is used to validate the model in this study. The inflow performance relationship IPR is a curve presented the well pressure ($p_{wf}$) as a function of production rate ($q$), and vice versa. The example of steady-state IPR is in Figure 5.7.

![Figure 5.7. IPR example](image)

Normally, IPR behaves as a straight line, when the well pressure is above the bubble point as in Figure 5.7 and exhibits the curvature profiles when the well pressure goes below the bubble point because the presence of gas in system. For the steady-state gas well under Darcy’s law, IPR exhibits the profile similar to the Figure 5.8.
5.3.1.3 Incompressible fluid model results. The IPR comparison between the FLUENT results of edge-drive model and the results of calculations utilizing the horizontal well equation (Eq. 5-26) are shown in Figure 5.9. The results from Eq. 5-26 presented in a unit of RVB/d, which obtained by assuming term B equal to 1.

Figure 5.8. IPR of gas well under Darcy’s law (Wang et al., 2009)
The FLUENT’IPR displays linear slope similar to the IPR of a well producing liquid. However, significant errors are evident when comparing the FLUENT generated IPR to the results from the horizontal well equation. Errors from the FLUENT’s results were verified and are presented in Table 5.2, along with the IPR data at each well pressure case. The typical errors of 26.10 % are observed in every case, which indicates the edge-drive model shall not be used as a representative of horizontal well model. The drainage geometry must be rectified to correct this error.
Due to the large error that occurred in validating the flow geometry of Augustine, it was determined that a new drainage geometry had to be defined to match the horizontal well equation defined by Joshi (1988).

5.3.1.4 New-proposed model. Joshi explained the horizontal well as the series of vertical wells next to each other drain the limited payzone thickness. With this explanation, the ends of horizontal well drained the reservoir with circular area with the rectangular drainage area at the center as per Figure 5.10.

### Table 5.2. Errors from the edge-drive model.

<table>
<thead>
<tr>
<th>$p_{wf}$ (psi)</th>
<th>q 300 ft FLUENT (RVB/d)</th>
<th>q 300ft equation (RVB/d)</th>
<th>% error from equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>1.948</td>
<td>2.636</td>
<td>26.10</td>
</tr>
<tr>
<td>500</td>
<td>1.723</td>
<td>2.332</td>
<td>26.10</td>
</tr>
<tr>
<td>800</td>
<td>1.498</td>
<td>2.027</td>
<td>26.10</td>
</tr>
<tr>
<td>1000</td>
<td>1.348</td>
<td>1.825</td>
<td>26.10</td>
</tr>
<tr>
<td>1200</td>
<td>1.199</td>
<td>1.622</td>
<td>26.10</td>
</tr>
<tr>
<td>1600</td>
<td>0.899</td>
<td>1.216</td>
<td>26.10</td>
</tr>
<tr>
<td>1800</td>
<td>0.749</td>
<td>1.014</td>
<td>26.10</td>
</tr>
<tr>
<td>2000</td>
<td>0.599</td>
<td>0.811</td>
<td>26.10</td>
</tr>
<tr>
<td>2200</td>
<td>0.450</td>
<td>0.608</td>
<td>26.09</td>
</tr>
<tr>
<td>2500</td>
<td>0.225</td>
<td>0.304</td>
<td>26.10</td>
</tr>
</tbody>
</table>
Figure 5.10. Top view concept of horizontal well by Joshi (1991)

The new-proposed model of 300 ft horizontal well is created in DesignModeler with the concept of symmetry plane. Unlike previous model, this model is no longer presented as continuous series of horizontal well. Thus, this model represents the complete 300 ft openhole horizontal well instead. In this new model, all outer boundaries of model are applied with the reservoir pressure. Figure 15.11 displays the new-proposed model based on Joshi’s concept.
The new-proposed model was validated with the incompressible fluid approach and its IPR curve is plotted in Figure 5.12 with the errors approximately 0.2 % from the horizontal well equation presented in Table 5.3.

Figure 5.12. IPR Comparison between FLUENT result of new-proposed model and horizontal well equation
Table 5.3. Error from the new-proposed model.

<table>
<thead>
<tr>
<th>( p_{\text{ref}} ) (psi)</th>
<th>q 300 ft (RVB/d)</th>
<th>q 300 ft equation (RVB/d)</th>
<th>% error from the equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>200</td>
<td>2.640</td>
<td>2.636</td>
<td>0.177</td>
</tr>
<tr>
<td>500</td>
<td>2.336</td>
<td>2.332</td>
<td>0.177</td>
</tr>
<tr>
<td>800</td>
<td>2.031</td>
<td>2.027</td>
<td>0.178</td>
</tr>
<tr>
<td>1000</td>
<td>1.828</td>
<td>1.825</td>
<td>0.180</td>
</tr>
<tr>
<td>1200</td>
<td>1.625</td>
<td>1.622</td>
<td>0.181</td>
</tr>
<tr>
<td>1600</td>
<td>1.219</td>
<td>1.216</td>
<td>0.185</td>
</tr>
<tr>
<td>1800</td>
<td>1.016</td>
<td>1.014</td>
<td>0.190</td>
</tr>
<tr>
<td>2000</td>
<td>0.813</td>
<td>0.811</td>
<td>0.196</td>
</tr>
<tr>
<td>2200</td>
<td>0.609</td>
<td>0.608</td>
<td>0.209</td>
</tr>
<tr>
<td>2500</td>
<td>0.305</td>
<td>0.304</td>
<td>0.263</td>
</tr>
</tbody>
</table>

Errors from the revised drainage model are <1% for incompressible flow. Hence the model was deemed to be the correct flow geometry and to be validated for the case of incompressible fluid flow.

5.3.2. Compressible Fluid Model Validation  In this section, the drainage geometry model validated using incompressible flow in section 5.3.1 is validated further using a natural gas as a fluid model. Prior to discussing this validation, the natural gas properties correlations in petroleum engineering, which are used to determine the properties in Equation 5-4 are introduced along with the equation of states used to determine the gas density in FLUENT.

5.3.2.1 Natural gas properties  Natural gas is a mixture of hydrocarbons. The properties of natural gas depend on its compositions. If the compositions are known, the physical properties can be obtained by several methods invented by natural gas engineers. This section briefly reviews natural gas properties, which are used in this study.

- Mole fraction. The mole fraction can be defined by (Ikoku, 1992):

  \[
  y_i = \frac{n_i}{\sum_{i} n_i}
  \]  

  (5-6)
where \( y_i \) is the mole fraction of component \( i \), \( n \) is the number of moles of component \( i \) and \( \Sigma n_i \) is the total moles in mixture/natural gas

- Apparent molecular weight. The concept of pseudo property helps characterizing the apparent molecular weight of a natural gas. The apparent molecular weight can be defined as (Ikoku, 1992):

\[
MW_g = \Sigma y_i MW_i
\]

(5-7)

where \( MW_g \) is the apparent molecular weight of natural gas and \( MW_i \) is the molecular weight of component \( i \). The molecular weight of natural gas’s components can be found in an Appendix A

- Real gas law. Natural gas in the reservoir is not in the ideal gas condition. The general equation that can approximate the real natural behavior of natural gas is (Wang et al., 2009):

\[
pV = ZnRT
\]

(5-8)

where \( p \) is the pressure (psi), \( V \) is the gas volume (ft\(^3\)), \( Z \) is the gas deviation factor or Z factor, \( R \) is the universal gas constant and equals to 10.73 psi ft\(^3\)/lb-mol-R and \( T \) is the absolute temperature (R), which can obtained from °F+460.

- Natural gas density. The density of natural gas can be defined as (Wang et al., 2009):

\[
\rho_g = \frac{pMW_g}{ZRT}
\]

(5-9)

where \( \rho_g \) is the natural gas density (lb/ft\(^3\))

- Pseudo critical properties. With the known gas compositions, the pseudo critical pressure of natural gas can be obtained from (Wang et al., 2009):
\[ p_{pc} = \Sigma y_i p_{ci} \] (5-10)

where \( p_{pc} \) is the pseudo critical pressure of natural gas (psi) and \( p_{ci} \) is the pseudo critical temperature of component \( i \) (psi).

In the similar manner, the pseudo critical pressure of natural gas can be obtain from (Wang et al., 2009):

\[ T_{pc} = \Sigma y_i T_{ci} \] (5-11)

where \( T_{pc} \) is the pseudo critical temperature of natural gas (R) and \( T_{ci} \) is the pseudo critical temperature of component \( i \) (R)

- Pseudo reduced properties. Pseudo reduced properties are simply the ratio between the gas actual properties and pseudo critical properties. The pseudo reduced pressure and temperature are (Wang et al., 2009):

\[ p_{pr} = \frac{p}{p_{pc}} \] (5-12)
\[ T_{pr} = \frac{T}{T_{pc}} \] (5-13)

Where \( p_{pr} \) is the pseudo reduced pressure and \( T_{pr} \) the pseudo reduced temperature

- Z factor correlation. The common method to determine Z factor of Standing and Katz (1942) requires users to read the chart (Figure 5.13) after the pseudo reduced pressure and temperature are obtained.
In order to avoid the errors due to chart reading, the Z factor correlation by Hall-Yarborough (1973) is used in the calculation of real gas law this study. The Hall-Yarborough method consists of the following equation (Ikoku, 1992):

\[ z = \frac{0.06125 p_p t e^{-1.2(1-t)^2}}{y} \]  

(5-12)

where \( t \) is the reciprocal of pseudo reduced temperature (\( T_{pc}/T \)) and \( y \) is the reduced density, which can be obtained from the solution of Equation 5-12 (Ikoku, 1992):
0 = -0.06125 p_{pr} t e^{-1.2(1-t)^2} + \frac{y + y^2 + y^3 - y^4}{(1 - y)^3} \\
- (14.76t - 9.76t^2 + 4.58t^3) y^2 \\
+ (90.7t - 242.2t^2 + 42.4t^3) y^{(2.14+2.82t)} \\ (5-13)

- Equation of state. In FLUENT, the application of Z factor does not exist as an input. The density of multi-component natural gas must be defined by the user. Hence, a module to calculate gas density by the equation of state had to be selected in this study. The Peng-Robinson equation of state was selected for the model.

Peng-Robinson equation of state (PR EOS) was introduced in 1976. The main objectives are to provide the reasonable Z factor near the critical point as well as should be applicable to all fluids in the natural gas processes (Nasri et al., 2009). It has been accepted to use widely in several application in the industry.

Peng-Robinson is a “cubic” equation. It requires three parameters: critical temperature (T_{cm}), critical pressure (p_{cm}) and acentric factor (\omega_m) of mixtures. In FLUENT, the critical temperature and pressure of mixture in Peng-Robinson equation use the mixing rule, which is based on one-fluid van der Waals mixing rules. Equations 5-13 and 5-14 define the critical temperature and pressure in Peng-Robinson equation, respectively (ANSYS FLUENT User’s Guide, 2011).

\[
T_{cm} = \frac{\left[\sum_{i} x_i \left(\frac{T_{ci}}{p_{ci}}\right)^{0.5}\right]^2}{\sum_{i} x_i \left(\frac{T_{ci}}{p_{ci}}\right)} \quad (5-14)
\]

\[
p_{cm} = \frac{T_{cm}}{\sum_{i} x_i \left(\frac{T_{ci}}{p_{ci}}\right)} \quad (5-15)
\]
The acentric factor of the mixture is simply defined by the mole mixing rule ([ANSYS FLUENT User’s Guide, 2011]):

\[ \omega_m = \sum y_i \omega_i \] (5-16)

where \( \omega_i \) is the acentric of gas component , which are available in an Appendix A.

Peng-Robinson equation is defined as (Nasri et al., 2009):

\[
p = \frac{RT}{V - b} - \frac{a_T}{[V(V + b) + b(V - b)]}
\] (5-17)

\[
a_T = a_c \alpha
\] (5-18)

\[
b = 0.07780 \frac{RT_{cm}}{p_{cm}}
\] (5-19)

\[
a_c = 0.45724 \frac{R^2T_{cm}^2}{p_{cm}}
\] (5-20)

\[
\alpha^{0.5} = 1 + m[1 - (\frac{T}{T_{cm}})^{0.5}]
\] (5-21)

\[
m = 0.37464 + 1.54226\omega_m - 0.26992\omega_m^2
\] (5-22)

where \( p \) is the absolute pressure(psia) \( T \) is the absolute temperature ( °R), \( V \) is the specific molar volume ( ft³/mole) and \( R \) is the universal gas constant

- Gas viscosity correlation. In this study the common gas viscosity correlation by Lee et al.(1966) is used. The gas viscosity (\( \mu_g \)) is defined by (Wang et al., 2009):

\[
\mu_g = K \exp (X\rho_g^\gamma)
\] (5-23)

\[
K = \frac{(9.4 + 0.02MW_g )T^{1.5}}{209 + 19MW_g + T}
\] (5-24)
\[ Y = 2.4 - 0.2X \]  
\[ X = 3.5 + \frac{986}{T} + 0.01MW_g \]

where the gas viscosity (\( \mu_g \)) is in a unit of centipoises (cp)

**5.3.2.2 Validation steps in the compressible fluid model.** In this section, the correct drainage geometry from the incompressible fluid model, obtained from section 5.4.4, is used with the natural gas. The final goal is to obtain an IPR relationship, as in the previous validations. The approach remains the same but there are some differences in terms of fluid properties and zone setup as per the following steps:

- Activate the multi-species function in FLUENT in Problem set up:Models
- Select the mixture-template. In Edit of Mixture Specie, select CH\(_4\) (Methane) and C\(_2\)H\(_6\) (Ethane). CH\(_4\) and C\(_2\)H\(_6\) must be copied from the FLUENT libraries before the mixture specie selection
- In Materials panel, at mixture-template, select the density model as Peng-Robinson. FLUENT shall perform the density calculation by Peng-Robinson equation of state (PR EOS).

Figure 5.14 illustrates the specie/ mixture set up.
The mixture calculations on the critical temperature, critical pressure and critical specific volume use the one fluid Van der Waals mixing law (Eq. 5-14, 5-15). The acentric factor uses the mole weighted mixing law (Eq. 5-16) as per Figure 5.15. The set up on mass diffusivity and thermal conductivity can be ignored because the assumption of non-diffused and isothermal system.
• Gas viscosity is assumed to be constant all over the domain in each well pressure case. In each case, the gas viscosity value is the arithmetic mean of the viscosity value at the outer boundary and well boundary obtained from the correlation in section 5.3.1 (Eq. 5-23, 5-24, 5-25 and 5-26). The average gas viscosity can be input in the mixture properties template. This value shall also be used in the horizontal well equation.

• In zone condition, the permeability setup can be performed in the similar manner as in the incompressible fluid setup. The material name must be set as the mixture template to specify the zone’s fluid as mixture. In Fixed values tab, users specify the mass fraction of C$_2$H$_6$. This setup constraints the constant gas compositions throughout the zone, which means no diffusion, occurs. Mass fraction of C$_2$H$_6$ is calculated by:

\[
Y_{C_2H_6} = \frac{y_{C_2H_6} MW_{C_2H_6}}{MW_g}
\]  

(5-27)
where $Y_{C_2H_6}$ is the mole fraction of C$_2$H$_6$ and MW$_{C_2H_6}$ is the molecular weight of C$_2$H$_6$.

Figure 5.16 illustrated Fixed values setup.

- The energy equation is now activated because PR EOS is used. To obtain the isothermal condition, where the energy equation is not computed, the fixed temperature in a zone is defined in Fixed values tab.

- Proceed with the simulation in the similar processes as in the incompressible model case. The volume flow rates are in a unit of MSCF/d. The volume flow rates at surface acquires from the conversion of mass flow rate results with the gas density (Eq.5-9) at surface condition: Pressure = 14.7 psi and temperature = 60°F.
5.3.2.3 Compressible fluid model results. The IPR comparison between the FLUENT’s results and the results from the horizontal equation (Eq. 5-4) with the compressible fluid (natural gas) are presented in Figure 5.17. The IPR curve from FLUENT exhibits the similar characteristic as the natural IPR well under Darcy’s law (Figure 5.8). Small errors gradually increase at larger drawdowns, as shown in Table 5.4.

Figure 5.17 IPR Comparison between FLUENT result and horizontal well equation: natural gas model
Table 5.4. Errors from the natural gas model

<table>
<thead>
<tr>
<th>$P_{wf}$ (psi)</th>
<th>$q_{300ft}$ equation (MSCF/d)</th>
<th>$q_{300ft}$ FLUENT (MSCF/d)</th>
<th>%error from equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>67.59</td>
<td>70.90</td>
<td>4.90</td>
</tr>
<tr>
<td>200</td>
<td>67.37</td>
<td>70.41</td>
<td>4.53</td>
</tr>
<tr>
<td>500</td>
<td>65.50</td>
<td>68.01</td>
<td>3.84</td>
</tr>
<tr>
<td>800</td>
<td>61.90</td>
<td>64.02</td>
<td>3.43</td>
</tr>
<tr>
<td>1000</td>
<td>58.57</td>
<td>60.45</td>
<td>3.22</td>
</tr>
<tr>
<td>1200</td>
<td>54.52</td>
<td>56.17</td>
<td>3.02</td>
</tr>
<tr>
<td>1500</td>
<td>47.16</td>
<td>48.48</td>
<td>2.79</td>
</tr>
<tr>
<td>1700</td>
<td>41.44</td>
<td>42.54</td>
<td>2.66</td>
</tr>
<tr>
<td>1800</td>
<td>38.34</td>
<td>39.34</td>
<td>2.59</td>
</tr>
<tr>
<td>2000</td>
<td>31.72</td>
<td>32.50</td>
<td>2.47</td>
</tr>
<tr>
<td>2400</td>
<td>16.82</td>
<td>17.21</td>
<td>2.30</td>
</tr>
</tbody>
</table>

The % errors shown in Table 5.4 are relatively small. The author identifies the causes of errors are:

- Different methods to define the gas compressibility: The Z-factor correlation by Hall-Yarborough (Eq. 5-12) was developed from the Starling-Carnahan equation of state (Dake, 1983) while the compressibility of natural gas in FLUENT utilizes the cubic equation of state PR-EOS. Those two methods are different by their individual numerical structures and tend to provide some deviations in their results.

- The Z factor in Equation 5.4 is the arithmetic mean of Z factor at the outer boundary wall and well boundary, obtained from the Z factor correlation Eq 5.12, while FLUENT assigns a value to each cell.

- Another cause of errors is the different in locality of compressibility in the equation and FLUENT model. In the natural gas horizontal well equation (Eq. 5-4), the compressibility factor is defined as the average compressibility between the outer boundary (at $r_e$) and the wellbore (at $r_w$). However, in FLUENT, the compressibility of natural gas is determined individually in each small discrete cell through the equation of state PR-EOS algorithm.
Errors occur in this compressible fluid model validation were not considered significant and the model validation was deemed to be satisfactory and ready to use in comparing the P-n-P completion to OHMS completions.
6. COMPLETION MODELS AND PROCEDURES

This section describes how a single transverse fracture intersecting P-n-P completion and OHMS completion were modeled included in the fundamental horizontal well model developed and verified in Chapter 5.

6.1. FRACTURE MODEL

Hydraulically induced fractures within unconventional reservoirs often yield complex fracture morphology. For this study, a single, uniform width hydraulic fracture has been assumed to simplify the problem. The concept of constant width fracture similar to Figure 1.6 is used in DesignModeler. A single transverse fracture is inserted in the center of the 300 ft horizontal well model. Figure 6.1 illustrates the single transverse fracture position in this study.

![Figure 6.1. Position of single transverse fracture](image)

In DesignModeler, the symmetry plane is assigned at the middle of the fracture. Thus, the propped fracture width \( w \) is modeled only in half. The fracture height is assumed to be equal to the formation thickness (60 ft). The rectangular sections of the formation are sub-divided into three parts to facilitate mesh generation. The middle section has its length in Y-direction equal to \( 2x_f \). The connectivity between the fracture
and the horizontal well shall be discussed in section 6.2 and 6.3. Figure 6.2 illustrates the transverse fracture in the horizontal well model from DesignModeler.

![Image of fracture in DesignModeler](image)

**Figure 6.2.** Transverse fracture in DesignModeler.

The fracture is assumed to be an isotropic porous media, where two horizontal permeability: X and Y direction, and the vertical permeability: Z direction, are all equal. The fracture permeability is determined from Equation 1-3 by known fracture conductivity (md-ft) and propped fracture width (inch). In FLUENT, the fracture permeability can be assigned into the bodies/zones of fractures in the similar way presented in Figure 5.8. Natural gas in fracture is assumed to be flowing under Darcy’s law.

### 6.2. PLUG-AND-PERF COMPLETION MODEL

The well completed with Plug-and-perf completion method (P-n-P) allows
natural gas flowing to the well through the perforation holes only. Previously, the perforation holes were the paths that hydraulic fractures were initiated from. So, the hydraulic fractures must connect directly to the perforation holes.

In reality, the fractures are created through the cluster of perforation holes, which display the complex fractured path or tortuosity. Due to the limitations of the meshing qualities and computational resources, these details are not modeled and included in this study. The fracture-well connectivity of P-n-P in this study is assumed to be two 180° perforation holes in the vertical (Z axis) direction connect with two separated fracture bodies: upper fractured body and lower fractured body. The perforation holes are in the perfect cylindrical shapes with the diameter of 0.44 inch based on the commercial fractured-gun performance data (Schlumberger, 2007). The suggested penetration depth of 1.5 times of well diameter, 9 inch, is used (Wutherich et al., 2012). The perforation holes and two fracture bodies in the –X directional view are displayed in Figure 6.3.

![Figure 6.3. Perforation holes and fracture bodies from the –X view](image)

The perforation holes are assigned as the non-porous zones. No crush zone and formation damage are around and at the end of perforation holes. These assumptions
allow natural gas from the formation interval to flow into the holes. The well pressures are assumed to act in normal direction at the end of perforation holes, which are the only outlets in this completion. Figure 6.4 illustrates the locations of outlets at perforation holes.

![Figure 6.4. Outlets of perforation holes](image)

The remaining boundary of the inner horizontal well is assumed to be a sandface behind the cased and cemented liner. The boundary condition of “wall” is used in FLUENT without any frictional losses occurring at the boundary wall. Figure 6.5 displays the sectional view of the no flow boundary behind the cased and cemented liner.
6.3. OPENHOLE MULTISTAGE COMPLETION MODEL

The hydraulic fracture in OHMS is assumed intersecting directly to the horizontal well. The fracture is divided into two parts: upper fracture body and lower fracture body, because the meshes are consistently distributed around the well bore more than the meshes from the single fracture body. The good-distributed meshes eliminate the reverse flow problems when the solver performs the iterations. Figure 6.6 displays the comparison of mesh distribution between single fracture model (left) and two fractures model (right).
Figure 6.6. Comparison of mesh distribution between 1 fracture model (left) and 2 fractures model (right).

Figure 6.7 illustrate the OHMS model. The right side is the magnified view of the well-fractures connections.

In OHMS, the openhole portion of the well is not cased and cemented. Thus, the natural gas flows through fracture bodies and openhole portion. The outlets consists of two
outlets from the fracture bodies and one outlet from the openhole section displayed in Figure 6.7. The constant well pressure ($p_{wf}$) is assumed to act in normal directions to all outlets.

All the horizontal sections without fractures are assumed to be the openhole portion. The openhole portion has no formation damage and is a perfect cylinder with a constant diameter. The openhole packer profiles are not included in this model. Figure 6.8 illustrates the sectional view of openhole portion of OHMS model.

![Figure 6.8 Sectional view of OHMS](image)

**6.4. SIMULATION AND PARAMETRIC STUDIES**

More than two hundred fifty simulations were performed comparing the P-n-P and OHMS completion methods. The simulation results are depicted as the relationship between dimensionless fracture conductivity, $C_{fd}$ (Eq. 1-4) and fold of increase, FOI (Eq. 1-5). The base case utilizes reservoir data from Table 5.1 and varies fracture conductivity (15,000; 13,000; 11,000; 9,000; 7,000; 5,000; 3,000, 1,000 and 500 md-ft) using ceramic proppant data (CARBO Ceramics Inc., 2007)
6.4.1. **Base Case.** The base case is the study based on the initial reservoir data from Table 5.1. The results are the relationships between the dimensionless fracture conductivity ($C_{fd}$) defined in Equation 1-4 and the folds of increase (FOI) defined in Equation 1-5. In order to vary the $C_{fd}$ variables, different values of fracture conductivity (15000, 13000, 11000, 9000, 7000, 5000, 3000, 1000 and 500 md-ft) are assigned to the model in FLUENT based on the long-term conductivity data from the commercial ceramic-coated proppant without the non-Darcy factors involved (CARBO Ceramics Inc., 2007). In P-n-P and OHMS models, a constant propped fracture width ($w$) of 0.01 inch and a fracture half-length ($x_f$) of 150 ft are assumed. All the simulation cases use a flowing bottomhole pressure ($p_{wf}$) of 1000 psi. The results are presented as two distinct curves from two different completion systems by X-axis is $C_{fd}$ and Y-axis is FOI of each completion type compares to the un-stimulated well. Results from base case simulation are also presented in section 8.1 by using the redefined relative conductivity term (Eq. 2-7) defined by Augustine (2011) instead of dimensionless conductivity term, to facilitate a comparison with Augustine’s results.

6.4.2. **Parametric Studies.** The parametric studies consist of:

6.4.2.1 **Parametric study case 1: fracture width ($w$).** The fracture width study uses the same reservoir conditions as the base case. The fracture conductivity range remains the same (md-ft to 15000 md-ft) The fracture half-length ($x_f$) is 150 ft. Two propped fracture width cases of 0.1 inch and 0.3 inch are simulated and compared with the base case :$w = 0.01$ inch. Fracture width is varied to determine whether the fracture width has an effect on the FOI of these two completion methods provided in section 8.2.1.

6.4.2.2 **Parametric study case 2: penetration ratio ($x_f/r_e$).** The penetration ratio study uses the same reservoir conditions as the base case. The propped fracture width ($w$) is kept as 0.01 inch. The fracture conductivity range remains the same (15000 md-ft to 500 md-ft). Two additional fracture half-length cases: 200 ft ($I_x = 0.67$) and ($I_x = 0.83$) are simulated and compared with the base case( $x_f = 150$ ft, $I_x = 0.5$ ). The study of penetration ratio was made to determine the effect of penetration ratio on the FOI of two completions. The discussion in section 8.2.2 provides the penetration ratio results in term of redefined relative conductivity and compare to the Augustine’s results.
6.4.2.3 Parametric study case 3: \( (k_v/k_h) \). The vertical to horizontal permeability ratio \( (k_v/k_h) \) study changes the value of vertical to horizontal permeability of reservoir from 0.1 in base case to 0.5 and 1.0. The remaining reservoir and fracture data are similar to the base case. The study on the reservoir’s vertical to horizontal permeability ratio \( (k_v/k_h) \) is to determine the effect of vertical to horizontal permeability ratio to FOI of two completions. Section 8.2.3 provides a discussion of the penetration ratio results in terms of redefined relative conductivity and compares those results to the results of Augustine (2011).
7. SIMULATION RESULTS

7.1. BASE CASE RESULTS

CFD simulation of the base case was performed according to the procedures discussed in section 6.4.1. The results, presented in this section as a relationship of the fold of increase (FOI = J/J₀) and the dimensionless conductivity (Cfd), are shown in Figure 7.1.

![Base case comparison; P-n-P and OHMS](image)

Figure 7.1. Base case results

Both curves of P-n-P and OHMS exhibit the same behavior as the fold of increases insignificantly increase when the dimensionless conductivities increases. The base case results show the folds of increases in natural gas production from the well completed with OHMS are average 8.2% higher than the fold of increase from the well completed with P-n-P method in the range of Cfd used in this study.
7.2. RESULTS FROM PARAMETRIC STUDIES

7.2.1. Results from the Parametric Study Case 1: Fracture Width (w). The parametric study on the propped fracture width was performed according to the procedures in section 6.4.2. Figure 7.2 and Figure 7.3 display the results from the fracture width study on the well completed with P-n-P and OHMS, respectively. Both Figure 7.2 and 7.3 display a small increase in FOI from the significantly greater propped fracture widths which is 10 times (0.1 inch) and 30 times (0.3 inch) from the base case width. Some lower \( C_{fd} \) values display the insignificant higher fold of increases than the fold of increases from the higher \( C_{fd} \) values due to the iterative errors.

![Parametric study case1: fracture width; P-n-P](image)

Figure 7.2. Results: parametric study case1: fracture width; P-n-P
Figure 7.3. Results: parametric study case1: fracture width; OHMS

The combination of Figure 7.2 and 7.3 becomes a comparison of the well complete with P-n-P and OHMS in the fracture width study, shown in Figure 7.4.
**Figure 7.4.** Results: parametric study case1: fracture width; P-n-P & OHMS

According to Figure 7.4, the increments in propped fracture width do not change the comparative relationships in term of production of the well completed with P-n-P and the well completed with OHMS, from the base case; \( w = 0.01 \) inch. The well completed with OHMS out-produces the well completed with P-n-P in both cases of \( w = 0.1 \) inch and 0.3 inch. The average increments of the well completed with OHMS to the well completed with P-n-P in case of \( w = 0.1 \) inch and 0.3 inch are 7.4 % and 7.9% respectively.
7.2.2. Results from the Parametric Study Case 2: Penetration Ratio ($x_f/r_e$).

The parametric study on the penetration ratio ($x_f/r_e$) was performed according to the procedure in section 6.4.3. Figure 7.5 and Figure 7.6 display the results from the penetration ratio study on the well completed with P-n-P and OHMS, respectively. According to Figure 7.5 and 7.6, the productions significantly increase with the higher penetration ratio. In the higher penetration ratio or higher fracture half-length cases, point out higher productivity with increased fracture length.

**Figure 7.5.** Results: parametric study case 2: penetration ratio; P-n-P
Figure 7.6. Results: parametric study case 2: penetration ratio; OHMS

The combination of Figure 7.5 and 7.6 becomes a comparison of the well completed with P-n-P and OHMS in the penetration ratio study, shown in Figure 7.7.
Increasing the penetration ratio does not change the comparative relationship in term of production of the well completed with P-n-P and wells completed with OHMS in the base case : \( I_x = 0.5 \). In both cases, where penetration ratio are 0.67 and 0.83, the well completed with OHMS out-produces the well completed with P-n-P by the average of 7.5% and 11.7 %, respectively.

### 7.2.1. Results from the Parametric Study Case 3: \( k_v/k_h \)

The parametric study of the vertical to horizontal permeability ration (\( k_v/k_h \)) was performed according to the procedure in section 6.4.4. Figure 7.8 and Figure 7.9 displays the results from the \( k_v/k_h \) ratio study on the well completed with P-n-P and OHMS, respectively. Both Figure 7.8 and 7.9 display the same trend when the \( k_v/k_h \) ratios of reservoir increase from the base case. With the higher reservoir’s \( k_v/k_h \) ratio, fold of increase decreases. This analysis was performed on a constant reservoir pay of 60 feet, which is a relatively thick formation for a horizontal well. \( k_v \) becomes more important in production from a thick reservoir versus
a thin one with a horizontal well. Had a reservoir thickness of 10 feet been used, the differences between $kv/kh=0.5$ and the other cases in Figure 7.8 would be much smaller.

![Parametric study case3: $k_v/k_h$; P-n-P](image)

Figure 7.8. Results: parametric study case3: $k_v/k_h$; P-n-P
The combination of Figure 7.8 and 7.9 becomes a comparison of the well complete with P-n-P and OHMS in reservoir's $k_v/k_h$ study, shown in Figure 7.10.
According to Figure 7.10, the higher reservoir’s $k_v/k_h$ ratio does not change the comparative relationship in term of production of the well completed with P-n-P and well completed with OHMS in the base case; $k_v/k_h=0.1$. In both cases, where $k_v/k_h$ are 0.50 and 1.0, the well completed with OHMS out-produces the well completed with P-n-P by the average of 13.2% and 15.5%, respectively.
8. RESULTS DISCUSSION

8.1. BASE CASE DISCUSSION

In general, varying fracture conductivity does not have a significant effect for either completion method, as shown by the curves in Figure 7.1. Both curves give almost the same values of fold of increase within the high range, right hand side, of conductivity with small decreases at the lowest range, left hand side. It can be concluded that using higher fracture conductivity is not a primary goal for fracturing design in low permeability reservoirs. The difference in term of production from two completion methods: P-n-P and OHMS need to be verified further using the contour analysis in section 8.1.1.

8.1.1. Contour Analysis. According to the results in section 7.1, the fold of increase from the well completed with OHMS is slightly greater than the fold of increase from the well completed with P-n-P. The reasons behind these results can be observed from the velocity contours from CFD-Post. Figure 8.1 displays the locations of two velocity contour planes: symmetry plane and mid-sectional plane, relative to the overall model.

Figure 8.1. Locations of velocity contour planes
8.1.1.1 The velocity contours of OHMS model. The velocity contour at mid-sectional plane from –Y directional view is in Figure 8.2 and the close-up contour near the fracture is in Figure 8.3.

Figure 8.2. Velocity contour at mid-sectional plane of OHMS model
Figure 8.3. Close-up view near fracture: velocity contour at mid-sectional plane of OHMS model

The velocity contour at the symmetry plane of OHMS model is in Figure 8.4 with the close-up view near the well in Figure 8.5.

Figure 8.4. Velocity contour at symmetry plane of OHMS model
According to Figure 8.2, the natural flow with the higher velocity, higher flow rate at the hydraulic fractured than at the openhole section. The flow rate is highest at the middle of fracture based on the velocity profile at Figure 8.3 and 8.5.

8.1.1.2 The velocity contours of P-n-P model. The velocity contour at mid-sectional plane and the close-up view at symmetry plane are in Figure 8.6 and 8.7, respectively.
Figure 8.6. Velocity contour at mid-sectional plane (-Y view) of P-n-P model
Figure 8.7 depicts the perforations and near wellbore area of the P-n-P completion, previously shown in Figure 6.3, but including the velocity contours. According to Figure 8.5 and 8.6, the perforation holes in P-n-P model act as the tunnels brought the natural gas from fracture and reservoir near fracture to the well. The high velocity spots occur at the connections of perforation holes and fractures. The velocities at the outlets of model, the only connections to the horizontal well, are higher than the velocities at the outlet of OHMS model.

The further analysis is to separate the mass flow rate from the fracture part and the openhole part of OHMS model and compares them with the mass flow from P-n-P model. Table 8.1, contains the mass flow rate of natural gas flow divided the flow from fracture and openhole section, with the mass flow rate from P-n-P model from base case. The data in table 8.1 were obtained directly from model. They must be multiplied by 2 to obtain the 300ft mass flow rate.

According to Table 8.1, the mass flow rates from fracture section in OHMS are slightly greater than the total mass flow rates from P-n-P model. Although the fracture in OHMS model directly intersects with the horizontal well and has the larger contact area
(Figure 8.3, 8.4 and 8.5) than the fracture-perforation contact area in P-n-P model, the higher velocities at the perforation holes of P-n-P models (Figure 8.6 and 8.7) compensate the disadvantage in term of fracture contact area of P-n-P model. This results in the closer fracture flow rates from P-n-P and OHMS model, shown in Table 8.1. On the other hand, the difference in total model flow rate comes from the openhole section of OHMS model, which is blocked by casing in P-n-P model. In the actual applications, where the formation damage in drilling appears, the openhole’s permeability is reduced and the openhole shall contribute only an insignificant flow rate. The total flow rate of the well completed with OHMS and P-n-P can be indifferent in this case.

Table 8.1. Mass flow rate from fracture and openhole sections in OHMS and overall flow from P-n-P: 150 ft results in base case

<table>
<thead>
<tr>
<th>OHMS (kg/s)</th>
<th>OHMS fracture flow (kg/s)</th>
<th>OHMS Openhole flow (kg/s)</th>
<th>P-n-P flow(kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0390</td>
<td>0.0370</td>
<td>0.0019</td>
<td>0.0362</td>
</tr>
<tr>
<td>0.0390</td>
<td>0.0370</td>
<td>0.0020</td>
<td>0.0362</td>
</tr>
<tr>
<td>0.0390</td>
<td>0.0370</td>
<td>0.0019</td>
<td>0.0362</td>
</tr>
<tr>
<td>0.0390</td>
<td>0.0370</td>
<td>0.0019</td>
<td>0.0361</td>
</tr>
<tr>
<td>0.0390</td>
<td>0.0370</td>
<td>0.0020</td>
<td>0.0361</td>
</tr>
<tr>
<td>0.0389</td>
<td>0.0370</td>
<td>0.0019</td>
<td>0.0361</td>
</tr>
<tr>
<td>0.0389</td>
<td>0.0370</td>
<td>0.0020</td>
<td>0.0360</td>
</tr>
<tr>
<td>0.0389</td>
<td>0.0369</td>
<td>0.0020</td>
<td>0.0357</td>
</tr>
<tr>
<td>0.0388</td>
<td>0.0368</td>
<td>0.0019</td>
<td>0.0352</td>
</tr>
</tbody>
</table>

8.1.2. Comparison of Base Case with Augustine’s Results. It is difficult to compare the results of this work to the work of Augustine (2011), including all of the parametric studies. Augustine’s work does not disclose all assumptions in the modeling method. However, a comparison has been made in this work to gain insight as to any differences in results of the current study compared to those of Augustine (2011). The intention of this comparison is not to compare the base case results to Augustine’s results quantitatively. Because there are too many differences in terms of the geometries, flow
assumptions and solver criteria, the comparison shall be performed qualitatively to
determine the agreements and disagreements between this study’s results and Augustine
(2011) results.

A comparison of this work to the results of Augustine were obtained through
digitizing Augustine’s FOI curve, over the portion of the curve covering the permeability
range for tight gas from the available publication. The Augustine’s case of $k_v/k_h = 0.1$, $L$
= 100 ft , $L/r_e = 0.5$ and $I_x = 0.5$ is selected to compare with the base case of this study.
Figure 8.8 displays the Augustine’s results of this selected case with the makers shown
the digitized range from the relative conductivity of 0.03 to 0.22. The arrow shows the
direction of the lower reservoir permeability.

![Figure 8.8. Augustine results case: $k_v/k_h = 0.1$, $L = 100$ ft , $L/r_e = 0.5$ and $I_x = 0.5$ with
the digitized range (Augustine, 2011)](image)

The base case data are converted to use Augustine’s redefined relative conductivity (Eq.
2-7). Figure 8.9 shows the results from base case compare to Augustine’s results.
According to Figure 8.8, both results from base case and from Augustine show an agreement in the same direction – OHMs completions outperform P-n-P completions. Considering the range of fold of increase obtained from this study using CFD models, both curves from two different completion methods give the fold of increases greater than one, which means both completion methods help improving gas production. However, in the results from Augustine, only the openhole completion provides the fold of increases greater than one, which improves the production from the un-stimulated well. The cemented completion (equivalent to P-n-P method in this study) provides the fold of increases less than one, which means reducing the gas production from the un-stimulated well. The author deems these results from cemented completion in Augustine’s work are unrealistic, since this P-n-P application has been proved to be commercially used widely in tight gas reservoirs. The author believes the over-simplified numerical models in
Augustine’s study did not represent the horizontal well with hydraulic fracturing completions correctly and leaded to these un-realistic outcomes.

8.2. DISCUSSION ON PARAMETRIC STUDIES

8.2.1. Discussion on Parametric Study Case 1: Fracture Width. According to Figure 7.3, 7.4 and 7.5, the higher fracture width does not significantly improve the fold of increase value, even though the 10 and 30 times higher in fracture width are used. This is to be expected as in all cases the fracture should perform as an infinite conductivity fracture, \( C_{\text{fd}} >> 30 \). Economides et al. (1994) gave an agreement as the propped fracture width is not the priority of hydraulic fracturing design in the low permeability reservoir. The parametric study in Augustine (2011)’s study on fracture width is not presented in his publication.

8.2.2. Discussion on Parametric Study Case 2: Penetration Ratio. According to Figure 7.5, 7.6 and 7.7, increasing in the penetration ratio (fracture half-length) significantly improves the fold of increase value, which agrees with Economides et al. (1994) as they stated the fracture half-length is a first priority of hydraulic fracturing design. The parametric study (Figure 8.10), by Augustine (2011) on the openhole model under the conditions of \( k_v/k_h=0.01 \), \( L = 100 \text{ ft} \) and \( L/r_e =0.5 \), shows the agreement on the low permeability reservoir range( right side).
Figure 8.10. Parametric study on penetration ratio on the openhole completion model (Augustine, 2011)

Figure 8.10 was digitized at the range of redefined relative conductivity of 0.02 to 0.25. Then, they were compared with the results from the parametric study on the penetration ratio mentioned in section 7.2.2 of OHMS model, shown in Figure 8.11. Augustine’s results show only the small improvements in the fold of increase from the increments in penetration ratio, which contrast to the results from this study.
8.2.3. Discussion on Parametric Study Case3: $k_v/k_h$. According to Figure 7.8, 7.9 and 7.10, increasing in reservoir’s $k_v/k_h$ reduce the fold of increase value. The reason is the reservoir loses the ability to produce vertically through the vertical permeability after transverse fracture was created. Transverse fracture drain the reservoir through the vertical permeability, which is normal to the fracture’s orientation. The natural gas production which is the denominator slightly increases when $k_v/k_h$ increases. On the contrary, the non-stimulated natural gas production, which is the denominator of fold of increase, increases dramatically compared to the nominator. This results in the lower fold of increase from the higher $k_v/k_h$. The results from Augustine (2011)’s study, in Figure 8.12, under the conditions of $L = 100$ ft, $I_x = 0.5$ and $L/r_e = 0.5$ also agree with this parametric study.
Figure 8.12. Parametric study on $k_v/k_h$ on the openhole and cemented completion models (Augustine, 2011)

The data in Figure 8.12 are digitized from the range of redefined from 0.007 to 0.27 and plotted in the same chart as the data from the $k_v/k_h$ study mentioned in section 7.2.1, shown in Figure 8.13.
Within the similar ranges of redefined relative conductivity, the results from Augustine’s study display the similar trend: increasing $k_v/k_h$ shall reduce the fold of increase. The data
from Augustine’s study display a steeper slope with the higher differences between the fold of increase from two completion models than the results from this study. The cemented completion results from Augustine give the fold of increase less than unity, which are not realistic. According to Figure 8.12, the fold of increase of cemented completion model will become zero at the very small magnitude of redefined relative conductivity. As the author mentioned in section 8.1.2, the results from Augustine cannot be quantitatively compared with the results from this study because of many different factors.
9. SUMMARY AND CONCLUSIONS

A study comparing two horizontal well completions—Plug-and-perf (P-n-P) and Open Hole Multi-stage System (OHMS)—in a tight gas reservoir was performed using Computational Fluid Dynamics (CFD) using steady state Darcy flow. The model developed in the study assumes a single constant-width transverse fracture placed at the center of 300 ft horizontal well model. The fracture height is assumed to be 60 ft, equal to the formation thickness. The model uses a representative tight gas reservoir, but does not include the presence of natural fractures or formation damage. The CFD horizontal well model was developed from the concept of edge-drive reservoir model used by Augustine (2011) and validated with the fundamental horizontal well equation by Joshi (1988). A pressure-based solver, laminar flow, isothermal conditions, constant gas viscosity and the gas compositional density model of Peng-Robinson equation of state were used in this CFD study.

Results are plotted as a graph of dimensionless conductivity versus folds of increase, created varying the fracture conductivity from 500 md-ft to 15000 md-ft. Results show that fold of increases from the well completed with OHMS are higher than the fold of increases from the well completed with P-n-P by approximately 8%, which is smaller than differences noted in the literature. Results from a parametric study of fracture width, fracture half-length and vertical to horizontal permeability ratio ($k_v/k_h$) show agreement that the well completed with OHMS slightly out produced the well completed with P-n-P.

The base case and parametric studies have been compared in the limited range of redefined relative conductivity with the results from Augustine (2011). These comparisons show agreements with the parametric studies by Augustine.

Conclusions of this work are as follows:

1. Historical comparisons of the production from P-n-P and OHMS completions systems are primarily based on production data, not reservoir modeling. While Augustine presents the first modeling effort, his work utilizes a 2-D linear model, which may not be an accurate representation of the problem.
2. CFD can be used to model near wellbore flow phenomena. FLUENT includes a porous media feature which has been utilized in this study.

3. Results of this work agree in the same direction as historical studies, i.e. OHMS completions outproduce P-n-P completions. However, this study finds a smaller production difference than historically reported in the literature.

4. It is difficult to compare results of this work to the results of Augustine, particularly extending into the high permeability range, because certain assumptions of his model are not known. Augustine demonstrates convergence of \( \frac{J}{J_0} \) to \( \sim 1 \) for OHMS, meaning that in a high permeability, conventional reservoirs, a fracture cannot significantly increase the ideal flow of an openhole. His results demonstrate \( \frac{J}{J_0} \) converges to near zero for P-n-P. The reasoning behind this is unclear, and is not explained his work.

5. Modeling the flow of natural gas through a fracture network in unconventional shales requires a more sophisticated approach than the model presented in this work.
10. FUTURE WORK

Natural gas flow in this study is assumed to flow under the Darcy’s law and simulated under the steady-state laminar flow solver. Further study should be made including the non-Darcy’s effect, transient flow or the turbulence model can be conducted in FLUENT.

The P-n-P transverse fracture model in this study is oversimplified because the actual hydraulic fractures, initiated by P-n-P method, display tortuosity near the wellbore. Tortuosity is a near wellbore phenomena, which occurs when a single fracture coalesces from several fracture starts in multiple perforation holes or when a single fracture turns to align with the far field stresses. The perforation modeling in this study does not include this and should be considered in the future.

The multi-transverse fracture study using CFD can be performed with the objective to optimize the fractures placement regarding to the natural gas production. By changing fracture spacing, the optimized distance of fracture placement can be determined. This is an important concern for industry and requires further study.

Heat transfer is not included in this study. Heat and thermal properties can be included in the CFD model to determine the actual effects from the heat transfer to the fluid flow system.

ANSYS Workbench has the capability to couple a mechanical structural solver with the thermal-fluid solvers. There are several problems related to multi-state fracturing in horizontal wells that require coupled fluid flow-geomechanical models. This problems include geo-mechanical failure of the formation during pump-in, sand production, well collapse, un-cemented liner failure due to hoop stress, and changes in fracture morphology due to changes in geo-mechanical properties. These problems can be evaluated using ANSYS workbench ‘coupling modules.

FLUENT is capable of simulating reactive flow and it has been used in the fluid flow related to chemical reactions for many years. The flow mechanism in shale gas, such as the gas desorbed mechanism, can be simulated in FLUENT using this reactive function.
APPENDIX A.
GAS PROPERTIES
GAS PROPERTIES FOR ALL CORRELATIONS AND SIMULATIONS:

FLUENT LIBRARY

Table A1 contains gas properties used in this study based on the FLUENT’s libraries.

Table A1 Gas components’ properties

<table>
<thead>
<tr>
<th>Gas components</th>
<th>Formula</th>
<th>Molecular Weight</th>
<th>Critical properties</th>
<th>Acentric factor(ω)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Methane</td>
<td>CH₄</td>
<td>16.043</td>
<td>343.338, 667.029</td>
<td>0.011</td>
</tr>
<tr>
<td>Ethane</td>
<td>C₂H₆</td>
<td>30.070</td>
<td>549.906, 706.624</td>
<td>0.100</td>
</tr>
</tbody>
</table>

Figure A1 and A2 displays the pure component properties of Methane in FLUENT.

![Image of Create/Edit Materials window showing properties for Methane](image)

Figure A.1. Methane’s properties in FLUENT-1.
Figure A.2. Methane’s properties in FLUENT-2.

Figure A3 and A4 displays the pure component properties of Ethane in FLUENT.

Figure A.3. Ethane’s properties in FLUENT-1.
Figure A.4. Ethane’s properties in FLUENT-2.
APPENDIX B.
CFD SIMULATION TUTORIAL.
CFD SIMULATION TUTORIAL

This tutorial explains how to setup the model to run the simulation in FLUENT. Users should have known the fundamental operations of FLUENT analysis prior reading the tutorial. The model of the well completed with OHMS method is used as an example.

1. Create the geometry domain in the DesignModeler. The procedures to create the geometry described in the user's guide (ANSYS DesignModeler User Guide, 2011).

2. After the domain’s geometry was created, it must be defined as “fluid”. The designate boundaries of domain must be specified at least one inlet and one outlet in order to be accepted by solver: FLUENT. In the OHMS model, 3 inlets, 2 outlets and 4 symmetry planes were defined as per Figure B.1. The remaining undefined boundaries shall be automatically defined as no-flow boundaries or “wall” in FLUENT.

3. The assignment can be performed by right click at the selected boundary. The green highlight depicts the selected surface/boundary. The name selection, Figure B.2, can be proceeding by typing the name desired by users. FLUENT automatically recognize the name of boundary by only the portion of the whole
given name, which must contains the keyword such as inlet or outlet. The example of boundaries’ name such as “pressureinlet-reg-1” contains the word “pressure”, which defines the specific type of inlet in FLUENT.

4. Before leaving the DesignModeler, users can check the gaps in domain by Tools→ Analysis Tools→ Fault Detection.

5. In the Meshing, the meshing setup on the global scale (whole domain) or the local (selected boundary or body). The global meshing controls are displayed in Figure B.3 with the details in Figure B.4. The local mesh control can be perform by select the surfaces/bodies of interest (green highlight indicated) → insert . Then
select the choice of local meshing control. The local mesh control is not used in this model.

Figure B.3. Location of global mesh control panel
6. According to Figure B.4, the meshing method is the “Automatic method” setting, which is used when there is no any specific methods are specified. The Automatic method identifies the swappable bodies and applies the sweep meshing. The remaining bodies are meshed with tetrahedral patch conforming method (ANSYS Lecture 4, 2012). The solver is specified as CFD: FLUENT.

Sizing is control by the relevance/relevance center. The definition of relevance/relevance center is according to Figure B.5. The 100 relevance is the finest scale. The setting of -20/medium was tested and used as the optimized values.
The inflation and patch confirming options are set to be default. The further global mesh control details, which are different from Figure B.4, are described in the ANSYS Meshing User’s Guide (2011).

7. The mesh qualities are available at the Mesh matric under the Statistics. The mesh qualities of fracture bodies are not in the recommended ranges but still can be used in FLUENT. The final mesh body is showed in Figure B.6.
Figure B.6. Geometry with meshes

8. Upon starting the FLUENT, select double precision option and use Parallel processing option: 10 processes setup is used for this model. Figure B.7 displays the initial setup for FLUENT.

9. Figure B.7. Initial setup of FLUENT

10. In FLUENT, the general set up is a steady state flow, pressure based solver and the absolute velocity formulation. The units can be adjusted to match with the desired inputs/outputs by clicking on the “Units…” and selecting the parameters and their units. The default unit in FLUNT is in SI unit. New units can be introduced by manually inputting the conversion factor (multiplying factor) and offset factor (shall be applied to the solutions after the conversion factor was multiplied to the SI unit). Figure B.8 displays the general setup with the unit setup. The mesh scale setup and quality checking can be performed on the panel above.
11. In model setup, the laminar vicious model is used. Energy equation is active (on), species transport is set as per Figure 5.10 in section 5.4.5. The complete model setups are in Figure B.9.
12. In Materials, select mixture-template and click on “Create/Edit…” . The density model of “real –gas-peng-robinson” is selected and users must click on “Change/Create” to active the density model. Gas mixture’s viscosity is input in the constant viscosity box. The mixture properties setup refers to Figure 5.11. After all material setups are completed, users must click on “Change/Create” again to record all setups. Figure B.10 displays the material setup panel.
13. In Cell Zone Conditions, each zone is defined the fluid inside, permeability and fixed value. The fluid in the “Material Name” box is set to be the mixture template. To activate the porous media and fixed values input, check on “Porous Zone” and “Fixed Values” boxes. The vicious resistance is related to the porous media’s permeability (Eq. 5-5). The fixed values setups see Figure 5.12. The porosity is not related to the velocity calculation because the “Superficial Velocity” is used for the porous formulation. The Operating Condition’s pressure can be left as a default 14.7 psi or set it as zero. Users must know the total pressure is the static pressure, from the calculation, combined with the operating pressure in here. Each zone must be set the properties according to the problem statements. Figure B.11 displays the cell zone setup.
14. In Boundaries Conditions, each particular boundary is assigned to fit with the problem statements. All inlets are set their “Gauge Total Pressure” and “Initial Gauge pressure” to 2800 psi with the pressure’s direction normal to the boundaries. The total temperature to 254 °F to obtain the isothermal condition. Figure B.12 illustrates the setup at inlet boundary setup.
At the outlets, the setup can be performed in the similar way as inlets, depicted in Figure B.13. All outlets use the gauge pressure 1000 psi with the direction normal to the boundary and set the backflow temperature to 254 °F.

At the walls, the no slip condition is used and set the temperature to 254 °F. The material's name is left as default because there is no heat transfer involves in this problem. Figure B.14 illustrates the wall setup.
15. In Solution Methods, the pressure-velocity Coupling is “SIMPLE” algorithm. The spatial discretization of gradient is Least Square Cell Based. The pressure gradient is set as “PRESTO!” which is recommended for the porous media domain. The pressure gradient “Standard” is an alternative option, if the solution hardly reaching the convergence. The remaining spatial discretization schemes are set as the “Second order Upwind”. Figures B.15 displays the solution methods setup.

![Solution Methods setup](image)

**Figure B.15. Solution methods setup**

16. In Solution Controls, the under-relaxation factors of pressure and momentum must be combined as 1 for the steady-state flow problems (ANSYS FLUENT User’s Guide, 2011). The remaining under-relaxation factors can be left as 1. Figure B.16 illustrates the under-relaxation factors setup. In Figure, B.16, both under-relaxation factors for pressure and temperature are 0.5, which provide the fastest convergence.
17. In Monitors, all residual sum are excluded for being the convergence criteria. Select “Residuals-Print,Plot” → Edit. Uncheck all marks in the convergence boxes of all governing equations. Figure B.17 illustrates the residual sums setup.

18. To monitor the mass conservation, where the true convergence reaches. Create two monitoring surface. First is the mass flow rate at all outlets. Another is the
overall mass flow rate in the domain. In Surface Monitors, select “Create”, then rename the surface of interest into the desired name. Set “Report Type” to “Mass Flow Rate”. Select all outlet boundaries as monitoring surfaces to obtain the mass flow rate at all outlets from the domain and select both inlets and outlets boundaries to be the monitoring surfaces of mass conservation. Lastly, check on “Plot” to display these monitoring values in each iteration. Figure B.18 displays the surface monitoring setup.

![Surface monitoring setup](image)

**Figure B.18. Surface monitoring setup**

19. In Solution initialization, users have several options to initialize the solution. By default, in the steady-state and laminar flow problems, user manual (ANSYS FLUENT User’s Guide, 2011) recommends the use the Hybrid Initiation; however it requires the considerable time to reach the true convergence. The models in this study, P-n-P and OHMS, reach the convergence faster by using the Standard Initialization. After select the Standard Initiation, select “Compute from”→ “all zones”. FLUENT will initialize the solution data from the prior settings from all fluid zones. Figure B.19 illustrates the solution initialization.
Figure B.19. Solution initialization

20. Before start iteration, mesh quality checking as well as mesh quality improvement can be performed by the manual keyword input. Users can type the input in the command window at the bottom of the screen. Type “mesh”→ press “Enter”. The available options will appear. By typing the keyword similar to the options, follow by pressing “Enter”, FLUENT performs the operation as per input. Figure B.20 displays the manual input on mesh.
21. The number of iterations can be set in the Run Calculation. Before the first time of iteration, FLUENT ask to perform “Check Case...” . Figure B.21 illustrates the Run Calculation setup.

![Run Calculation Setup](image)

Figure B.21. Run calculation setup
22. The iteration starts after users press “Calculate”. The convergence criteria are described in section 5.4.4 by observing the true mass conservation. While the iteration is running, users can stop it by press “Cancel” at the Working panel.

23. In Reports, users have a choice to obtain the report of data after the final iteration. “Fluxes” is an option to report the mass/heat flow of the desired boundaries. After the boundaries are selected, users press “Compute” to show the results. The properties at the desired surfaces/volumes can be obtained in the similar manners as in “Fluxes” from “Surfaces Integrals” and “Volume Integrals”, respectively. Figure B.22 displays the Reports.

![Figure B.22. Reports setup](image)

24. The post-processor in FLUENT can display the graphical results such as contours, particle tracks or the animation. The contours can be displayed by select “Contours” in “Graphics”. Then, specify the property and surfaces. Select “Display” to show the contour. Figure B.23 displays the contour of static pressure.
Figure B.23. Post-processor setup: Contour of static pressure
APPENDIX C.

RAW DATA AND DIGITIZED DATA.
RAW DATA AND DIGITIZED DATA

Raw data: section 5.5.3
Table C.1 contains the actual mass flow rate of 150ft half-model from FLUENT.

Table C.1. Raw data from FLUENT in section 5.5.3

<table>
<thead>
<tr>
<th>$P_{wf}$ (psi)</th>
<th>150 ft (half-model) natural gas mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>0.008181</td>
</tr>
<tr>
<td>200</td>
<td>0.008125</td>
</tr>
<tr>
<td>500</td>
<td>0.007848</td>
</tr>
<tr>
<td>800</td>
<td>0.007388</td>
</tr>
<tr>
<td>1000</td>
<td>0.006976</td>
</tr>
<tr>
<td>1200</td>
<td>0.006482</td>
</tr>
<tr>
<td>1500</td>
<td>0.005594</td>
</tr>
<tr>
<td>1700</td>
<td>0.004909</td>
</tr>
<tr>
<td>1800</td>
<td>0.004539</td>
</tr>
<tr>
<td>2000</td>
<td>0.003750</td>
</tr>
<tr>
<td>2400</td>
<td>0.001986</td>
</tr>
</tbody>
</table>

Raw data: section 7.1
Table C.2 contains the raw data of 150ft model in the base case from FLUENT

Table C.2. Raw data from FLUENT in section 5.5.3

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>P-n-P half-model mass flow rate (kg/s)</th>
<th>OHMS half-model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.018105</td>
<td>0.019481</td>
</tr>
<tr>
<td>13000</td>
<td>0.018080</td>
<td>0.019480</td>
</tr>
<tr>
<td>11000</td>
<td>0.018078</td>
<td>0.019488</td>
</tr>
<tr>
<td>9000</td>
<td>0.018070</td>
<td>0.019479</td>
</tr>
<tr>
<td>7000</td>
<td>0.018064</td>
<td>0.019477</td>
</tr>
<tr>
<td>5000</td>
<td>0.018050</td>
<td>0.019475</td>
</tr>
<tr>
<td>3000</td>
<td>0.018022</td>
<td>0.019468</td>
</tr>
<tr>
<td>1000</td>
<td>0.017850</td>
<td>0.019438</td>
</tr>
<tr>
<td>500</td>
<td>0.017608</td>
<td>0.019393</td>
</tr>
</tbody>
</table>
Raw data section 7.2.1

Table C.3 contains the raw data from 150ft-P-n-P model in section 7.2.1.

Table C.3. Raw data of P-n-P model from FLUENT in section 7.2.1

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>w=0.01 inch model mass flow rate (kg/s)</th>
<th>w=0.1 inch model mass flow rate (kg/s)</th>
<th>w=0.3 inch model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.018105</td>
<td>0.018321</td>
<td>0.018466</td>
</tr>
<tr>
<td>13000</td>
<td>0.018080</td>
<td>0.018299</td>
<td>0.018438</td>
</tr>
<tr>
<td>11000</td>
<td>0.018078</td>
<td>0.018288</td>
<td>0.018421</td>
</tr>
<tr>
<td>9000</td>
<td>0.018070</td>
<td>0.018265</td>
<td>0.018445</td>
</tr>
<tr>
<td>7000</td>
<td>0.018064</td>
<td>0.018264</td>
<td>0.018423</td>
</tr>
<tr>
<td>5000</td>
<td>0.018050</td>
<td>0.018230</td>
<td>0.018403</td>
</tr>
<tr>
<td>3000</td>
<td>0.018022</td>
<td>0.018205</td>
<td>0.018365</td>
</tr>
<tr>
<td>1000</td>
<td>0.017850</td>
<td>0.018006</td>
<td>0.018209</td>
</tr>
<tr>
<td>500</td>
<td>0.017608</td>
<td>0.017739</td>
<td>0.017956</td>
</tr>
</tbody>
</table>

Table C.4 contains the raw data from OHMS model in section 7.2.1.

Table C.4. Raw data of 150ft-P-n-P model from FLUENT in section 7.2.1

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>w=0.01 inch model mass flow rate (kg/s)</th>
<th>w=0.1 inch model mass flow rate (kg/s)</th>
<th>w=0.3 inch model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.019481</td>
<td>0.019546</td>
<td>0.019834</td>
</tr>
<tr>
<td>13000</td>
<td>0.019480</td>
<td>0.019539</td>
<td>0.019829</td>
</tr>
<tr>
<td>11000</td>
<td>0.019488</td>
<td>0.019533</td>
<td>0.019819</td>
</tr>
<tr>
<td>9000</td>
<td>0.019479</td>
<td>0.019526</td>
<td>0.019811</td>
</tr>
<tr>
<td>7000</td>
<td>0.019477</td>
<td>0.019525</td>
<td>0.019808</td>
</tr>
<tr>
<td>5000</td>
<td>0.019475</td>
<td>0.019521</td>
<td>0.019808</td>
</tr>
<tr>
<td>3000</td>
<td>0.019468</td>
<td>0.019531</td>
<td>0.019808</td>
</tr>
<tr>
<td>1000</td>
<td>0.019438</td>
<td>0.019502</td>
<td>0.019776</td>
</tr>
<tr>
<td>500</td>
<td>0.019393</td>
<td>0.019457</td>
<td>0.019728</td>
</tr>
</tbody>
</table>
Raw data section 7.2.2

Table C.5 contains the raw data from P-n-P model in section 7.2.2.

Table C.5. Raw data of 150ft-P-n-P model from FLUENT in section 7.2.2

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>$I_x = 0.5$ model mass flow rate (kg/s)</th>
<th>$I_x = 0.67$ model mass flow rate (kg/s)</th>
<th>$I_x = 0.83$ model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.018105</td>
<td>0.022889</td>
<td>0.029369</td>
</tr>
<tr>
<td>13000</td>
<td>0.018080</td>
<td>0.022879</td>
<td>0.029360</td>
</tr>
<tr>
<td>11000</td>
<td>0.018078</td>
<td>0.022875</td>
<td>0.029355</td>
</tr>
<tr>
<td>9000</td>
<td>0.018070</td>
<td>0.022866</td>
<td>0.029347</td>
</tr>
<tr>
<td>7000</td>
<td>0.018064</td>
<td>0.022854</td>
<td>0.029335</td>
</tr>
<tr>
<td>5000</td>
<td>0.018050</td>
<td>0.022833</td>
<td>0.029314</td>
</tr>
<tr>
<td>3000</td>
<td>0.018022</td>
<td>0.022777</td>
<td>0.029257</td>
</tr>
<tr>
<td>1000</td>
<td>0.017850</td>
<td>0.022487</td>
<td>0.028914</td>
</tr>
<tr>
<td>500</td>
<td>0.017608</td>
<td>0.022075</td>
<td>0.028329</td>
</tr>
</tbody>
</table>

Table C.6 contains the raw data from OHMS model in section 7.2.2.

Table C.6. Raw data of 150ft-OHMS model from FLUENT in section 7.2.2.

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>$I_x = 0.5$ model mass flow rate (kg/s)</th>
<th>$I_x = 0.67$ model mass flow rate (kg/s)</th>
<th>$I_x = 0.83$ model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.019481</td>
<td>0.024458</td>
<td>0.032666</td>
</tr>
<tr>
<td>13000</td>
<td>0.019480</td>
<td>0.024457</td>
<td>0.032664</td>
</tr>
<tr>
<td>11000</td>
<td>0.019488</td>
<td>0.024455</td>
<td>0.032661</td>
</tr>
<tr>
<td>9000</td>
<td>0.019479</td>
<td>0.024453</td>
<td>0.032657</td>
</tr>
<tr>
<td>7000</td>
<td>0.019477</td>
<td>0.024451</td>
<td>0.032650</td>
</tr>
<tr>
<td>5000</td>
<td>0.019475</td>
<td>0.024446</td>
<td>0.032638</td>
</tr>
<tr>
<td>3000</td>
<td>0.019468</td>
<td>0.024433</td>
<td>0.032610</td>
</tr>
<tr>
<td>1000</td>
<td>0.019438</td>
<td>0.024372</td>
<td>0.032472</td>
</tr>
<tr>
<td>500</td>
<td>0.019393</td>
<td>0.024282</td>
<td>0.032269</td>
</tr>
</tbody>
</table>
Raw data section 7.2.3

Table C.7 contains the raw data from the un-stimulated model used in section 7.2.3.

<table>
<thead>
<tr>
<th>Reservoir $k_v/k_h$</th>
<th>Un-stimulated-150ft model (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.006929</td>
</tr>
<tr>
<td>0.5</td>
<td>0.010564</td>
</tr>
<tr>
<td>1</td>
<td>0.012089</td>
</tr>
</tbody>
</table>

Table C.8 contains the raw data from P-n-P model in section 7.2.3.

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>$k_v/k_h=0.1$ model mass flow rate (kg/s)</th>
<th>$k_v/k_h=0.5$ model mass flow rate (kg/s)</th>
<th>$k_v/k_h=1.0$ model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.018105</td>
<td>0.018119</td>
<td>0.018138</td>
</tr>
<tr>
<td>13000</td>
<td>0.018080</td>
<td>0.018109</td>
<td>0.018120</td>
</tr>
<tr>
<td>11000</td>
<td>0.018078</td>
<td>0.018109</td>
<td>0.018114</td>
</tr>
<tr>
<td>9000</td>
<td>0.018070</td>
<td>0.018109</td>
<td>0.018114</td>
</tr>
<tr>
<td>7000</td>
<td>0.018064</td>
<td>0.018095</td>
<td>0.018109</td>
</tr>
<tr>
<td>5000</td>
<td>0.018050</td>
<td>0.018083</td>
<td>0.018094</td>
</tr>
<tr>
<td>3000</td>
<td>0.018022</td>
<td>0.018048</td>
<td>0.018060</td>
</tr>
<tr>
<td>1000</td>
<td>0.017850</td>
<td>0.017880</td>
<td>0.017893</td>
</tr>
<tr>
<td>500</td>
<td>0.017608</td>
<td>0.017634</td>
<td>0.017646</td>
</tr>
</tbody>
</table>

Table C.9 contains the raw data from P-n-P model in section 7.2.3.
Table C.9. Raw data of 150ft-OHMS model from FLUENT in section 7.2.3.

<table>
<thead>
<tr>
<th>Fracture conductivity (md-ft)</th>
<th>( k_v/k_h = 0.1 ) model mass flow rate (kg/s)</th>
<th>( k_v/k_h = 0.5 ) model mass flow rate (kg/s)</th>
<th>( k_v/k_h = 1.0 ) model mass flow rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15000</td>
<td>0.019481</td>
<td>0.020408</td>
<td>0.020835</td>
</tr>
<tr>
<td>13000</td>
<td>0.019480</td>
<td>0.204078</td>
<td>0.020838</td>
</tr>
<tr>
<td>11000</td>
<td>0.019488</td>
<td>0.020406</td>
<td>0.020837</td>
</tr>
<tr>
<td>9000</td>
<td>0.019479</td>
<td>0.020405</td>
<td>0.020836</td>
</tr>
<tr>
<td>7000</td>
<td>0.019477</td>
<td>0.020403</td>
<td>0.020833</td>
</tr>
<tr>
<td>5000</td>
<td>0.019475</td>
<td>0.020397</td>
<td>0.020828</td>
</tr>
<tr>
<td>3000</td>
<td>0.019468</td>
<td>0.020370</td>
<td>0.020804</td>
</tr>
<tr>
<td>1000</td>
<td>0.019438</td>
<td>0.020330</td>
<td>0.020765</td>
</tr>
<tr>
<td>500</td>
<td>0.019393</td>
<td>0.020300</td>
<td>0.020765</td>
</tr>
</tbody>
</table>

**Digitized data from Augustine in section 8.2.1**

Table C.10 contains the digitized data from the openhole completion model by Augustine used in section 8.2.1.

Table C.10. Digitized data from openhole model by Augustine, used in section 8.2.1.

<table>
<thead>
<tr>
<th>Redefined relative conductivity</th>
<th>( J/J_0 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2197</td>
<td>1.1077</td>
</tr>
<tr>
<td>0.1529</td>
<td>1.0851</td>
</tr>
<tr>
<td>0.1177</td>
<td>1.0736</td>
</tr>
<tr>
<td>0.0947</td>
<td>1.0623</td>
</tr>
<tr>
<td>0.0740</td>
<td>1.0508</td>
</tr>
<tr>
<td>0.0562</td>
<td>1.0393</td>
</tr>
<tr>
<td>0.0433</td>
<td>1.0384</td>
</tr>
<tr>
<td>0.0319</td>
<td>1.0375</td>
</tr>
</tbody>
</table>

Table C.11 contains the digitized data from the cemented completion model by Augustine used in section 8.2.1.
**Table C.11.** Digitized data from cemented completion model by Augustine, used in section 8.2.1.

<table>
<thead>
<tr>
<th>Redefined relative conductivity</th>
<th>$J/J_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2337</td>
<td>0.4554</td>
</tr>
<tr>
<td>0.1699</td>
<td>0.3689</td>
</tr>
<tr>
<td>0.1309</td>
<td>0.3253</td>
</tr>
<tr>
<td>0.1053</td>
<td>0.2711</td>
</tr>
<tr>
<td>0.0847</td>
<td>0.2384</td>
</tr>
<tr>
<td>0.0643</td>
<td>0.1840</td>
</tr>
<tr>
<td>0.0518</td>
<td>0.1513</td>
</tr>
<tr>
<td>0.0422</td>
<td>0.1399</td>
</tr>
<tr>
<td>0.0330</td>
<td>0.1285</td>
</tr>
<tr>
<td>0.0262</td>
<td>0.1064</td>
</tr>
</tbody>
</table>

**Digitized data from Augustine in section 8.2.2**

Table C.12 contains the digitized data the openhole completion model by Augustine used in section 8.2.2.

**Table C.12.** Digitized data from openhole completion model by Augustine, used in section 8.2.2.

<table>
<thead>
<tr>
<th>$I_x = 0.25$</th>
<th>$I_x = 0.5$</th>
<th>$I_x = 0.75$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Redefined relative conductivity</strong></td>
<td><strong>$J/J_0$</strong></td>
<td><strong>$J/J_0$</strong></td>
</tr>
<tr>
<td>0.258</td>
<td>1.151</td>
<td>0.243</td>
</tr>
<tr>
<td>0.201</td>
<td>1.117</td>
<td>0.193</td>
</tr>
<tr>
<td>0.163</td>
<td>1.106</td>
<td>0.135</td>
</tr>
<tr>
<td>0.122</td>
<td>1.062</td>
<td>0.094</td>
</tr>
<tr>
<td>0.095</td>
<td>1.062</td>
<td>0.063</td>
</tr>
<tr>
<td>0.068</td>
<td>1.050</td>
<td>0.047</td>
</tr>
<tr>
<td>0.048</td>
<td>1.039</td>
<td>0.032</td>
</tr>
<tr>
<td>0.032</td>
<td>1.016</td>
<td>0.022</td>
</tr>
<tr>
<td>0.023</td>
<td>1.016</td>
<td>0.015</td>
</tr>
</tbody>
</table>
Digitized data from Augustine in section 8.2.3

Table C.13 contains the digitized data the openhole completion model by Augustine used in section 8.2.3.

Table C.13. Digitized data from openhole completion model by Augustine, used in section 8.2.3.

<table>
<thead>
<tr>
<th>$k_r/k_h$=0.001</th>
<th>$k_r/k_h$=0.01</th>
<th>$k_r/k_h$=0.1</th>
<th>$k_r/k_h$=1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Redefined relative conductivity</td>
<td>$J/J_0$</td>
<td>Redefined relative conductivity</td>
<td>$J/J_0$</td>
</tr>
<tr>
<td>0.281</td>
<td>1.634</td>
<td>0.240</td>
<td>1.237</td>
</tr>
<tr>
<td>0.210</td>
<td>1.505</td>
<td>0.196</td>
<td>1.194</td>
</tr>
<tr>
<td>0.153</td>
<td>1.409</td>
<td>0.146</td>
<td>1.161</td>
</tr>
<tr>
<td>0.110</td>
<td>1.333</td>
<td>0.113</td>
<td>1.140</td>
</tr>
<tr>
<td>0.077</td>
<td>1.247</td>
<td>0.087</td>
<td>1.118</td>
</tr>
<tr>
<td>0.057</td>
<td>1.194</td>
<td>0.058</td>
<td>1.075</td>
</tr>
<tr>
<td>0.041</td>
<td>1.151</td>
<td>0.047</td>
<td>1.054</td>
</tr>
<tr>
<td>0.032</td>
<td>1.108</td>
<td>0.036</td>
<td>1.054</td>
</tr>
<tr>
<td>0.024</td>
<td>1.075</td>
<td>0.029</td>
<td>1.032</td>
</tr>
</tbody>
</table>

Table C.14 contains the digitized data the cemented completion model by Augustine used in section 8.2.3.
Table C.14. Digitized data from cemented completion model by Augustine, used in section 8.2.3.

<table>
<thead>
<tr>
<th>$k_r/k_h$</th>
<th>0.001</th>
<th>0.01</th>
<th>0.1</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$J/J_0$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.281</td>
<td>1.043</td>
<td>0.335</td>
<td>0.763</td>
<td>0.325</td>
</tr>
<tr>
<td>0.226</td>
<td>0.914</td>
<td>0.247</td>
<td>0.613</td>
<td>0.254</td>
</tr>
<tr>
<td>0.174</td>
<td>0.774</td>
<td>0.204</td>
<td>0.548</td>
<td>0.199</td>
</tr>
<tr>
<td>0.132</td>
<td>0.656</td>
<td>0.157</td>
<td>0.462</td>
<td>0.151</td>
</tr>
<tr>
<td>0.108</td>
<td>0.570</td>
<td>0.125</td>
<td>0.409</td>
<td>0.108</td>
</tr>
<tr>
<td>0.071</td>
<td>0.398</td>
<td>0.098</td>
<td>0.355</td>
<td>0.076</td>
</tr>
<tr>
<td>0.055</td>
<td>0.323</td>
<td>0.074</td>
<td>0.258</td>
<td>0.058</td>
</tr>
<tr>
<td>0.038</td>
<td>0.258</td>
<td>0.054</td>
<td>0.194</td>
<td>0.044</td>
</tr>
<tr>
<td>0.027</td>
<td>0.194</td>
<td>0.045</td>
<td>0.183</td>
<td>0.034</td>
</tr>
</tbody>
</table>


Lohoefer D., Snyder D.J., Seal R., 2010, “Long-Term Comparison of Production Results from Open Hole and Cemented Multi-Stage Completions in Barnett
Shale,” IADC/SPE Asia Pacific Drilling Technology Conference and Exhibition. IADC/SPE 136196.


Unconventional Resources Conference and Exhibition held in Vienna, Austria, 20-22 March 2012. SPE 150949.


VITA

Viriyah Theppornprapakorn was born in January, 1986, in Bangkok, Thailand. He received his bachelor degree in Mechanical Engineering from Chulalongkorn University, Bangkok, Thailand in 2008. He joined Foster Wheeler Cooperation, Thailand in July 2008 - September 2010 as a Piping Design Engineer. His duties there included preparing the piping and equipment arrangement drawings and the piping fabrication drawings for the refinery and petrochemical plant construction projects. Viriyah joined Inventa Technologies Pte., Singapore, as a Technical Assistant in February 2011-July 2011. He was assigned to provide the Front-End Engineering Design on piping and equipment, and provide the piping material specifications for the Carotene isolation plant, Thailand.

Viriyah came to Missouri University of Science and Technology in August, 2011 for Master of Science in Chemical Engineering and transferred to the Petroleum Engineering program in January, 2012. He worked under Dr. Shari Dunn-Norman as her research assistant. His thesis topic and thesis explores a comparison of plug and perf versus open hole completion methods in horizontal well multi-stage fracturing using CFD analysis. He received a Master of Science degree in Petroleum Engineering from Missouri University of Science and Technology in December 2013.