University of Stavanger

Faculty of Science and Technology

MASTER'S THESIS

Study program/ Specialization: Industrial Economics; drilling and project management	Spring semester, 2011 Open
Writer: Maiken Ravndal	………………………………………… (Writer's signature)
Faculty supervisor: Kjell Kåre Fjelde, UiS External supervisor(s):	
Title of thesis: **Models for Dynamic Kill of Blowouts**	
Credits (ECTS): 30	
Key Words: - Blowout - Dynamic kill - The AUSMV scheme - Numerical, two-phase modeling	Pages: 89 + enclosure: 33 Stavanger, ……………….. Date/year

Abstract

Blowouts represent one of the largest risks oil companies are exposed to during exploration and exploitation of petroleum resources, and the increasing complexity of e.g. drilling operations states the need for more sophisticated well control modeling tools.

Dynamic kill is a technique combining the static head of a kill fluid with frictional pressure losses in order to suppress the reservoir pressure and kill the blowout. The technique can be modeled dynamically with the use of two-phase flow simulators based on conservation equations and closure laws for the different flowing phases. The result is a better understanding of blowout scenarios and parameters like bottom hole pressure, flow rate, pump rate, mud volume and kill fluid density.

In this thesis a transient two-phase model based on the Advection Upstream Splitting Method (AUSMV) scheme has been constructed for the simulation of a dynamic kill. The starting point was a simple two-phase code for gas and liquid extended to a dynamic kill model suitable for oil and liquid. The extension was done through several programming steps: A numerical approach for finding the pressure parameter was implemented and verified, the friction model was changed to suit an oil blowout in a vertical well, a model for the productivity index (PI) was used to simulate the inflow of oil, the velocity of sound for the mixture fluid was changed according to recommendations, and finally a frictional pressure gradient for the relief well was implemented in order to find the required pump pressure for the dynamic kill operation. After running several simulations with different kill rates and observing the results, the model proved to be sufficient in describing a dynamic kill operation for a simplified oil blowout.

The simulations revealed that the model contained a stiff source term, the PI model, switching on/off on a short timescale creating oscillations in the curve. To avoid the oscillations the model required a more sophisticated time integration procedure involving small time steps far below the CFL (Courant-Friedrichs-Lewy) criterion. In addition the blowing well was found to be slightly friction dominated, the increase in friction was larger than the reduction of the hydrostatic pressure when oil displaced water in the well.

Future recommendations for the model include the implementation of several phases and the opportunity to simulate kill fluids with different densities. Wellbore geometries, blowout configurations and models for flow regimes and transitions should also be implemented. This will make the model more versatile for different dynamic kill scenarios in the future.

Table of Contents

Acknowledgement

This master thesis marks the end of my master degree at the University of Stavanger in Industrial Economics with specialisations in drilling and project management.

First of all I would like to thank my supervisor Professor Kjell Kåre Fjelde for providing me with an interesting and challenging master thesis related to my specialisation in drilling. I am very grateful for his help and guidance and his availability, he was available for help and discussion at any time, every day and every week.

I would also like to thank to my family and friends for their support through my years of study.

Stavanger, June 2011

……………………………………………

Maiken Ravndal

List of Figures

List of Tables

Nomenclature

AUSMV - Advection Upstream Splitting Method
BHP - Bottom Hole Pressure
BOP - Blow Out Preventer
CFL - Courant-Friedrichs-Lewy Condition
COMASim - Cherokee Offshore, MMS, Texas A&M Simulator
DHSV - Down Hole Safety Valve
FVS - Hybrid Flux-Vector Splitting Scheme
GOR - Gas Oil Relation
HHP - Hydraulic Horsepower
ID - Inner Diameter
IPR - Inflow Performance Curves
MD - Measured Depth
MSL - Mean Sea Level
OD - Outer Diameter
OLF - The Norwegian Oil Industry Association
PI - Productivity Index
PSA - Petroleum Safety Authority
PVT - Pressure Volume Temperature
TVD - True Vertical Depth
U.S - United States
U.S GoM OCS - United States Gulf of Mexico Outer Continental Shelf
UBD - Under Balanced Drilling
UK - United Kingdom

1 Introduction

1.1. Background

Blowouts represent one of the largest risks oil companies are exposed to during exploration and exploitation of petroleum resources. The consequences are huge as seen from the blowout in the Macodo well located in the Gulf of Mexico[1]:

On the 20th of April 2010 hydrocarbons escaped from the Macondo well up to the Transocean`s drilling rig Deepwater Horizon. The blowout resulted in explosions and fire causing the loss of 11 lives, the injury of 17 people and the history's largest spill of hydrocarbons.

The blowout was an outcome from chains of events resulting in an uncontrolled leakage of hydrocarbons from the well. Several attempts to seal the well was performed, but with no success. The well kept on leaking for a total of 87 days until the 19th of September 2010 when BP finally reported that the Macondo well was completely shut-in, both in casing and annulus. Approximately 4.2 million barrels of crude oil[2] had then been released to the sea causing huge environmental damage.

Figure 1-1 - Deepwater Horizon Located in the Gulf of Mexico, Mississippi Canyon Block 252 [15]

[1] This section is based on reference [3].
[2] Estimated number is taken from reference [1].

Drilling operations today are becoming more and more complex as the industry is moving into new, more challenging areas. Harsh, vulnerable environments and deepwater drilling is only a few of the challenges companies have to face in order to reach the valuable hydrocarbon reserves. The increasing complexity states the need for more sophisticated well control modeling tools, and among them software tools modeling blowouts and dynamic kill of blowouts. The tools are essential in planning and contingency procedures to predict any risks and minimize or avoid accidents like the Macondo blowout.

1.2. Study Objective

The main objectives of this thesis are to give a review of models for dynamic kill of blowouts, and program and implement a simple dynamic kill model to demonstrate the functionality and behavior of a dynamic, two-phase model for a dynamic kill operation.

1.3. Structure of the Thesis

The thesis is divided into 7 chapters. Chapter 1 gives the introduction which covers the background, study objective and structure of the thesis. The reader is also provided with background information relevant for the following chapters. Chapter 2 gives a theoretical review of the dynamic kill operation and method, while chapter 3 discusses two-phase models and dynamic modeling in detail. Available software models are presented in chapter 4, and a simple dynamic well kill model is programmed in chapter 5 to provide a comprehension of the dynamic kill process, the modeling and the results. Chapter 6 discusses uncertainties in dynamic modeling with focus on the transient model in chapter 5, and finally chapter 7 gives a summary of the thesis through conclusion and future recommendations.

1.4. Classification of Blowouts

A blowout can be defined as an uncontrolled flow of reservoir fluids (oil, gas, salt water) into the wellbore or to surface [4], and is usually a result of a kick; a flow of formation fluids into the wellbore failed to be controlled by the rig personnel.

For offshore operations blowouts can be classified as [6]:

- Surface blowout - Characterised by fluid flowing from the reservoir zone and up to the rig floor. Often associated with large-scale fires and most often a result from events during drilling, completion, production or workover.

- Underground blowout - Fluid is flowing from one formation zone to another with the use of the wellbore as fluid path. Often the fluid path goes from deep high pressure zones to shallower low pressure zones. Underground blowouts can either occur during drilling or in rare cases develop in completed wells. The first case is normally related to improper handling of a kick while the second case may occur due to improper cementing of casing, causing fluid flow; failure in casing due to tectonic movements or bad choice of casing steel quality [5].

- Subsea blowout - The blowout is located below sea level and exits the well at seabed. This type of blowout may extend over a large area causing problems with buoyancy for vessels and pollution to the sea. The subsea blowout is mostly caused by events during drilling and is very difficult to kill when occurring in deep seawaters.

1.5.Kicks and Blowout Causes

This section is based on reference [5] unless otherwise stated.

The first sign of a potential blowout is usually a kick and if not treated right it may lead to a blowout. Normally blowouts are caused by chain of events rather than a by one single event only, meaning that several barriers need to fail before a potential blowout can occur.

SINTEF [35] has developed along with several major oil companies a database recording offshore blowouts from January the 1st 1980 to January the 1st 2008. A total of 237 blowouts/well releases from the U.S Gulf of Mexico Outer Continental Shelf (U.S GoM OCS) and the Norwegian and United Kingdom (UK) waters was combined and divided into blowouts/well releases occurring in different operational phases. As one may see from Table 1-1, offshore blowouts/well releases mostly occurred during the phases development drilling and exploration drilling [13].

AREA	Dev. drlg	Expl. drlg	Unk. drlg	Comp-letion	Work-over	Production		Wire-line	Un-known	Total
						External cause*	No ext. cause*			
US GoM OCS	53	50		12	35	6	10	2	5	173
	30.6%	28.9%	0.0%	6.9%	20.2%	3.5%	5.8%	1.2%	2.9%	100.0%
UK, and Nor-wegian waters	9	31	2	6	9	1	2		4	64
	14.1%	48.4%	3.1%	9.4%	14.1%	1.6%	3.1%	0.0%	6,3	100.0%
Total	62	81	2	18	44	7	12	2	9	237
	26.2%	34.2%	0.8%	7.6%	18.6%	3.0%	5.1%	0.8%	3.8%	100.0%

* External causes are typical; storm, military activity, ship collision, fire and earthquake.

Table 1-1 - Number of Blowouts/Well Releases Experienced during Different Operational Phases [13].

Blowouts during drilling occur more often than blowouts in other phases. Planning a drilling operation requires great attention in trying to evaluate potential hazards that may occur and the effect they may have on costs, humans and the environment.

In the following, the most common causes for kicks and potential blowouts are reviewed [5].

1.5.1. Insufficient Mud Weight

If the hydrostatic pressure of the mud column is lower than the pressure of the reservoir fluids in the surrounding formation, reservoir fluids may enter the well inducing a kick.

1.5.2. Tripping

During drilling it is necessary to pull the drill string to e.g. change the drill bit. If the drill pipe is pulled out too quickly a suction pressure can occur resulting in a swab effect which reduces the pressure in the well and possibly inducing a kick and a potential blowout. This can also occur if the hydrostatic pressure is lost when pushing the drill string back into the well again. If the drill string is pushed in too quickly the formation may fracture inducing a kick.

1.5.3. Improper Hole Fill Up

In addition to the risk of taking a kick during tripping, a kick can also be induced if the hole is not properly filled with mud when pulling the drill string. The volume in the wellbore annulus decreases when the drill pipe is pulled which means that the bottom hole pressure (BHP) is reduced leaving the well in an underbalanced condition.

1.5.4. Disconnection of Riser

Special attention should be paid if the riser must be disconnected from the Blow Out Preventer (BOP) due to e.g. bad weather. In this situation it is very important to keep the well stable. The stability of the well is maintained by displacing the well up to the BOP with a heavy kill fluid e.g. brine, to keep a sufficient overbalanced hydrostatic pressure when the rig is disconnected.

1.5.5. Shallow Gas Zones

Shallow gas zones are gas zones penetrated before the BOP and surface casing has been installed [8]; when drilling the overburden layers of the well the BOP is useless because the overburden layer is too thin and weak and cannot handle a shut-in pressure. Shallow gas zones can in worst case create a cratered blowout, and oil companies do everything to avoid this by performing seismic studies and pilot hole drilling. If drilling through shallow gas zones is unavoidable in order to reach the deeper reservoir zones, an option may be subsea diversion where gas is directly displaced to the sea.

1.5.6. Blowout in Cased Wells

When dealing with cased wells it is important to distinguish between two conditions: When there is mud in the well and when there are reservoir fluids in the well. A cased and mud filled well gives a minor chance for a blowout if the mud is isolated from the formation with e.g. a plug. This is also valid for a mud column with an established overbalanced pressure. If in worst case a blowout should occur it is usually due to human errors like poor cementing or natural causes like casing failure due to tectonic movements.

When a completed well contains reservoir fluids in the wellbore the well control is depended by shut-in devices like the Christmas three and the down-hole safety valve (DHSV). If a blowout occurs in this situation it is most likely caused by independent accidents e.g. damage of the downstream equipment.

The production casing may also experience some kind of failure due to bad steel quality or erosion during production. If this happens hydrocarbons will leak into the annulus and build up a pressure that the protective casing may not withstand. Worst case scenario will be loss of well control and shut-in of the well at surface.

1.6.Controlling Kicks

There are several warning signs notifying a possible kick [5]:

- Increasing flow rate return.
- Pit volume gain.
- Well flowing with the pumps turned off.
- Decrease in pump pressure.
- Drilling break (increased penetration rate).
- Gas, oil or water in the mud.
- Change in drill string weight.
- Change in pressure.

If a kick is taken, the BOP will be closed and the well will be shut-in until the pressure has stabilised. The kick will then be circulated out through the choke line with a constant circulation rate, while the BHP is kept constant above the formation pressure by proper choke adjustments. There are 3 variations of this method [9]:

- Wait and weight method - Kill fluid is pumped into the well at the same time as the kick is being circulated out. The advantage is that the well is killed in one circulation.

- Driller's method - The kick will be circulated out before a heavier kill mud is injected. This method requires at least two complete circulations to ensure that the kick is circulated out and the well is stabilised.

- Concurrent method - Circulation begins immediately while the mud density is gradually increased when pumped down the well.

1.7. Killing Blowouts

If a kick fails to be controlled and results in a blowout different intervention methods can be used to kill the blowout and regain well control. The following are based on references [5], [6] and [10].

- Capping – The blowout is stopped mechanically by shut-in from surface; the flow paths exit point is closed. The method was one of the first used to control blowouts and can be described as a quick method mostly used for surface blowouts in shallow waters. The simplest equipment for capping consists of a pipe either fitted with a ball valve or blind rams together with a diverter line. Once the equipment is stabbed into the well the valve or ram is closed and the well is shut-in. A large pressure will then develop inside the well and pressure monitoring is therefore important to maintain full well control. The capping method requires access to the wells exit point and can therefore not be used for subsea blowouts.

- Momentum kill/Bullheading – A surface intervention method used to forcibly pump water, mud or brine into the annulus or the drill pipe to balance the reservoir pressure and kill the well statically. Water is normally pumped in first followed by heavy mud or brine. A requirement for the method is to have sufficient pumping capacity on board in order to push the reservoir fluids back. The advantage with the momentum kill/bullheading method is that the drill string does not need to go all the way down in the well. A disadvantage is that the drill crew does not know where the fluids will flow thus it can potentially cause an underground blowout.

- Bridging – Choking or opening up a restriction in the well may promote bridging which results in caving-in of the borehole, thus stopping the blowout. Induced bridging is not a well-used method due to the fear of losing the well.

- Cement – Fast reacting cement can be injected into the well once it is static to ensure full well control. When using this method it is important to evaluate the formation and set the plug in a way that reduces the possibility of formation fracturing.

- Depletion of the reservoir with relief wells – The blowout can be killed by depletion; production of reservoir fluids from nearby wells reducing the pressure in the reservoir.

- Dynamic kill – A blowout control technique using a relief well to pump kill fluid into a blowing well. The kill fluid creates a frictional pressure drop which in addition to the hydrostatic pressure exceeds the pressure in the formation and kills the well. Dynamic kill can also be used in combination with capping intervention; the capping stack is used as a choke to give additional backpressure. The dynamic kill method will be reviewed in detail in Chapter 2.

If a blowout has occurred it is very important to prevent ignition and a potential fire in developing. This can be done with water supply from the fire pumps at the platform or from assisting vessels. The water drops reduces the threat of ignition, emulsifies the oil making it harder to ignite as well as cools-off equipment and structures.

1.8. Blowout Consequences

The consequences of a blowout are huge, and affect many different aspects:

- Environmental damage and harm to the environment.
- Injury or in worst case loss of lives.
- Loss of valuable hydrocarbon reserves and income.
- Unexpected costs; blowout control costs and clean-up costs.
- Rig and equipment damage.
- Loss of the operator and personals credibility.
- Delayed time-schedule for further drilling in the surrounding area.

1.9.Well Control – Required Barriers

Well control is "*a collective expression for all measures that can be applied to prevent uncontrolled release of wellbore effluents to the external environment or uncontrolled underground flow* [7]". In relation to kick and blowout, it is very important to establish and ensure well control. This can be done through barriers, safety systems and backup systems to prevent that a single event leads to a blowout.

In Norway the petroleum industry is controlled and supervised by the Petroleum Safety Authority (PSA). The supervision embraces a number of activities which gives a basis for deciding whether companies operating on the Norwegian continental shelf are fulfilling their responsibility and operate acceptably in all phases of the industry. Notices of orders are given if laws and regulations are not fulfilled [17].

In addition to the laws and regulations given by the PSA, the NORSOK standard (developed by the Norwegian petroleum industry) is recommended to be followed during petroleum activities.

According to NORSOK Standard D-010 [7], Rev. 3 2004 section 4.2.3.2, the following shall be satisfied when establishing well barriers:

"There shall be one well barrier in place during all well activities and operations, including suspended or abandoned wells, where a pressure differential exists that may cause uncontrolled cross flow in the wellbore between formation zones.

There shall be two well barriers available during all well activities and operations, including suspended or abandoned wells, where a pressure differential exists that may cause uncontrolled outflow from the borehole/well to the external environment [7]"

The NORSOK requirements ensure that barriers are in place for both flow between the wellbore and the surrounding formation thus avoiding a potential underground blowout; and between the wellbore and the external environment preventing a surface - or subsea blowout.

Identifying and establishing well barriers in different operations is the main element to achieve full well control. In a blowout situation the barrier elements can be divided into three groups [5]:

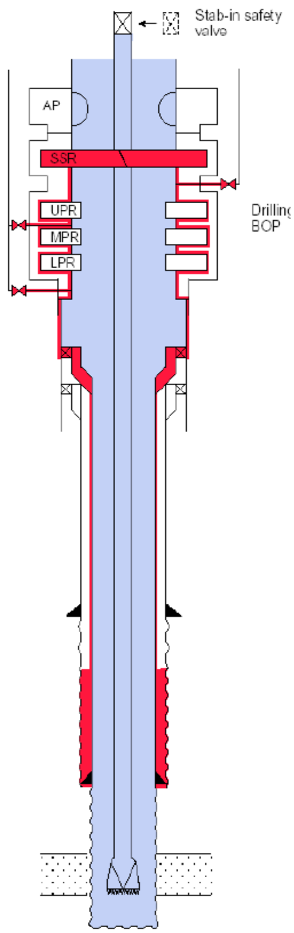
- *"Passive Blowout Barriers:*
    - *Open hole formation*
    - *Casing*
    - *Packer*
    - *Tubing*
    - *Wellhead body*

- *Fluid Blowout Barriers:*
    - *Mud*
    - *Work-over and completion fluids*
    - *Packer fluids*

- *Shut-in Blowout Barriers:*
    - *DHSV*
    - *BOP*
    - *Christmas three*
    - *Lubricator*
    - *Plugs*
    - *Check valves [5]"*

The fluid column together with the BOP is the main barrier elements during drilling, completion and work-over operations, see Figure 1-2.

5.8.1 Drilling, coring and tripping with shearable drill string



Well barrier elements	See Table	Comments
Primary well barrier		
1. Fluid column	1	
Secondary well barrier		
1. Casing cement	22	
2. Casing	2	Last casing set.
3. Wellhead	5	
4. High pressure riser	26	If installed.
5. Drilling BOP	4	

Note

None

Figure 1-2 - Well Barrier Schematic [7]

2 Dynamic Kill

2.1 The Dynamic Kill Operation

The following is based on references [6], [10], [11], [12] and [14].

Dynamic kill is a technique combining the static head of a kill fluid with frictional pressure drop in order to control and kill blowouts. The technique was developed, and formalized by Mobil Oil Corporation in 1978 during the surface blowout in the Arun gas field in Indonesia: The time-period for the killing operation was expected to be one year but with the use of the dynamic kill technique the well was successfully killed after 89 days with only one relief well. Through the years the technique has proven to be a successful blowout control method used on several blowouts around the world, both in the Gulf of Mexico [50] and in the Norwegian sector [51].

The technique can either use a relief well or surface intervention e.g. injection string to enter the wellbore. When using surface intervention major pressure will be lost inside the work-string due to the small diameter and the high kill rate. This makes it difficult to achieve adequate friction in the annulus. Compared to a relief well, when circulating through the annulus, less frictional pressure drop and pumping power will be required for the same kill rate. Relief wells are therefore usually chosen as entering method: Two s-shaped relief wells are drilled on each side of the blowing well where one serves as a contingency.

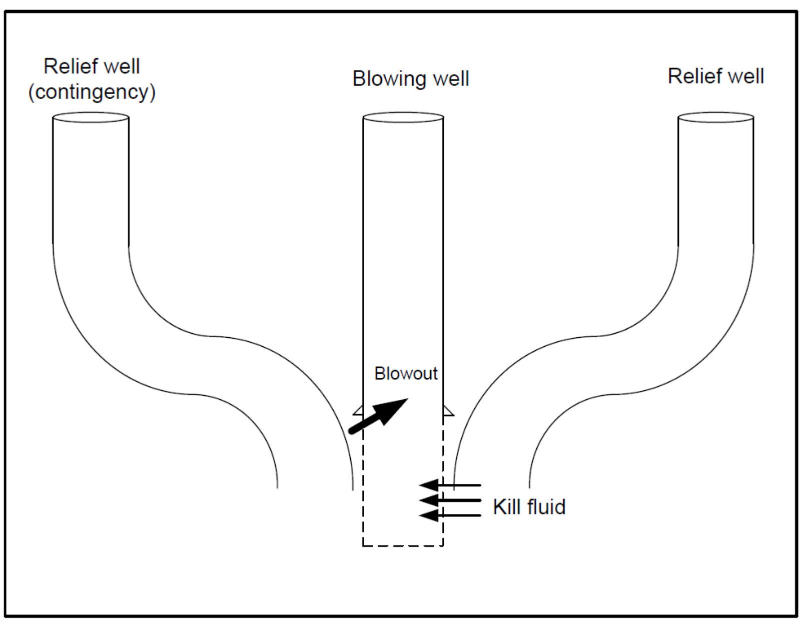


Figure 2-1 - Relief Wells

After the relief wells are drilled the next step is to establish a communication link between the wellbores of the blowing well and the relief well. This is a critical factor; if flow connection is not verified the kill operation is impossible. The relief well can either intersect the wellbore at the well bottom or it can intersect the wellbore at a distance above the blowing formation depending on where the blowout is located.

The dynamic kill procedure starts by pumping a light kill fluid, normally salt water when offshore, down the kill and choke lines of the relief well and either up through the annulus or up through the drill string of the blowing well. The objective with using a lighter kill fluid is to ensure a stable communication between the two wells without breaking down the formation and loosing fluid to a fractured part of the formation. The water itself does not have enough static head to control the inflow but by using high pump rates an additional frictional pressure is created in the well. The frictional pressure supplements the hydrostatic pressure of the kill fluid creating a pressure overbalance which stops the inflow. This can be shown by the following simple equation:

$$P_{Hydr} + P_{wh} + \Delta P_{fric} > P_R$$

Where: P_{Hydr} - hydrostatic pressure of the fluid column P_{wh} - pressure at wellhead, ΔP_{fric} - frictional pressure and P_R - average static reservoir pressure.

Once the blowout is under control and the inflow from the reservoir has stopped water is too light to hold the well dead for a longer period of time. Heavier kill mud is therefore introduced to fully stabilise the well. During the transition from light to heavy kill fluid it may be necessary to stepwise increase the density in order to maintain well control during the operation. The final kill fluid controlling the well statically normally has a density no heavier than the mud used before the blowout took place. This is to ensure that the pressure in the well stays within the fracture gradient and pore pressure window to avoid fracturing the well after the dynamic kill operation is finished.

The dynamic kill concept is schematically shown in Figure 2-2 below: The blowout profile of a blowing gas well is sketched to the left in the diagram. When a kill fluid is injected into a blowing well the fluid itself does not have enough hydrostatic pressure to kill the well. However, with sufficient pump rate frictional pressures are created increasing the pressure in the well to above the reservoir pressure thus killing the well.

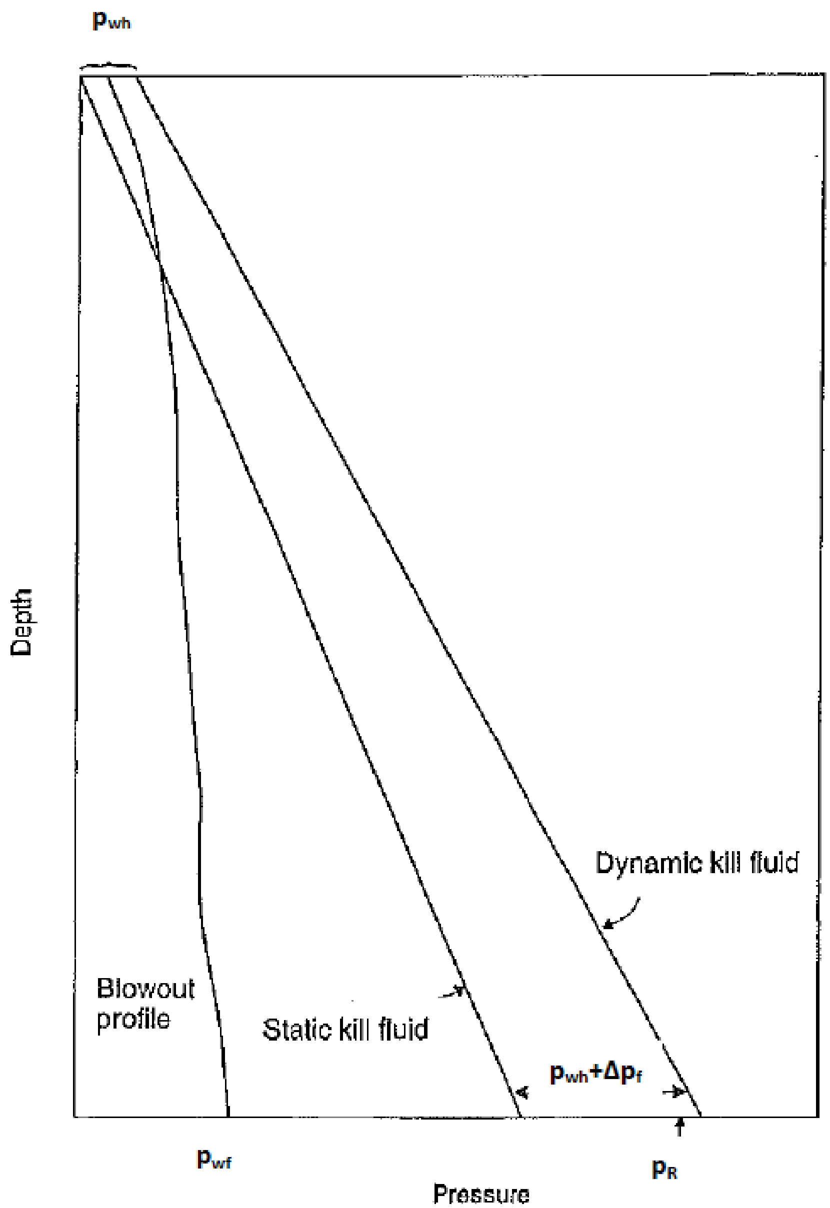


Figure 2-2 - Dynamic Kill Concept [27]

The dynamic kill technique is a good choice in blowout situations involving high flow rates, hard access locations and wells with damaged wellhead equipment. It is also a good technique to use in wells with low fracture gradients due to the ability to control the dynamic kill to stay within the fracture and pore-pressure window.

2.2 The Dynamic Kill Method

In this chapter two ways of determining the minimum dynamic kill rate and relevant dynamic kill parameters will be presented.

2.2.1. Dynamic Kill Parameters

To carry out a dynamic kill operation several parameters must be predetermined. E.M Blount and E. Soeiinah [14] proposed a way to determine the necessary parameters for a dynamic kill without the use of complex mathematics and software models:

- Kill fluid density:

  The density of the kill fluid can be determined by finding the fluid which will increase the frictional pressure component with the same amount as the hydrostatic pressure component is reduced, when introducing a bubble of gas into a single phase flow.

  The total pressure in the well is the sum of the hydrostatic weight of the fluid, and the frictional pressure:

  $$P\,[psi] = \Delta P_{hyd}[psi] + \Delta P_f[psi] \qquad (2.1)$$

  The frictional pressure can be represented by the following equation:
  $\Delta P_f[psi] = \frac{CfL\,\rho_l V_l^2}{d_h}$
  Where: C is a constant, f is the Fanning friction factor, $\rho_l\,[ppg]$ is the density of the kill fluid, $V_l\,[ft./s]$ is the velocity of the kill fluid and $d_h[inches]$ is the hydraulic diameter.

  A gas bubble is then assumed to enter the liquid flow, giving a total velocity of $V = \frac{V_l}{(1-\phi_g)}$, where ϕ_g is the fraction of the gas volume.
  With the introduction of the gas bubble, the density of the fluid can be written as: $\rho_f[ppg] = \rho_l(1-\phi_g) + \rho_g\phi_g$

  By deriving equation (2.1) with respect to the gas fraction $\frac{dP}{d\phi_g} = \frac{\mathrm{d}\Delta P_{hyd}}{\phi_g} + \frac{\mathrm{d}\Delta P_f}{\phi_g}$, and rearranging the terms the density of the kill fluid can be calculated with the following equation:

$$\rho_f [ppg] = \frac{12{,}836\ Ps\ [psi]}{TVD\ [ft.]} \tag{2.2}$$

Here: P_S $[psi]$ is the static formation pressure and *TVD [ft.]* is the true vertical depth of the blowing well. The factor 12,836 is a conversion factor giving the density in *pounds per gallon (ppg)*.

- Kill fluid injection rate:

The total frictional pressure losses during injection of kill fluid are given by the following equation:

$$\Delta P_f\ [psi] = \frac{11{,}4\ fL\ \rho_f\ q^2}{d_e^5} \tag{2.3}$$

Where:

- f is the Fanning friction factor: $f\ = \frac{0{,}25}{\left(2\ log\frac{d_h}{e}+1{,}14\right)^2}$, with the hydraulic diameter defined by $d_h[inches] =\ d_0 - d_i$ and e $[inches]$ as the pipe`s roughness.
- L $[ft.]$ is the measured depth of the blowing well, ρ_f $[ppg]$ is the density of the kill fluid and q $[bpm]$ is the flow rate of the kill fluid.
- The diameter, d_e^5 $[inches]$, is the equivalent pipe diameter when the kill fluid is pumped through the annulus: $d_e^5 =\ (d_0 - d_1)^3(d_0 + d_1)^2$

The frictional pressure loss in the blowing well is given by:

$$\Delta P_{f_b}[psi] =\ P_s\ -\ \Delta P_{hyd} \tag{2.4}$$

Here the BHP of the blowing well corresponds to $P_s[psig]$ – the static formation pressure.

Setting the expression for the frictional pressure in equation (2.3) equal to the expression in equation (2.4), and solving for the flow rate the following equation is achieved for the kill fluid rate up the blowing well:

$$q_b[bpm] = \left[\frac{(P_s - \Delta P_{hyd})}{11{,}41\ \rho_f}\left(\frac{d_e^5}{fL}\right)_b\right]^{1/2} \tag{2.5}$$

The fractional leak-off is given by $(1-k)$ where $k = {}^{q_b}/_{q_r}$ is the fraction of the flow entering the wellbore; q_b $[bpm]$ – flow up blowout well (kill rate), q_r $[bpm]$ – injection down relief well. Multiplying the expression for k into equation (2.5), gives the required injection rate down the relief well:

$$q_r[bpm] = \frac{1}{k}\left[\frac{(P_s-\Delta P_{hyd})}{11{,}41\,\rho_f}\left(\frac{d_e^5}{fL}\right)_b\right]^{1/2} \tag{2.6}$$

- Size of the relief well:

The expression for the frictional pressure loss is given by equation (2.7):

$$\Delta P_f[psi] = C\left(\frac{fL}{d_e^5}\right)\rho_f q^2 \tag{2.7}$$

The parameters in equation (2.7) are described previously.

If complete turbulence is assumed the "flow resistance" $\left(\frac{fL}{d_e^5}\right)$ is independent of fluid properties and can for a well divided into N sections be expressed as: $\left(\frac{fL}{d_e^5}\right)_{total} = \sum_{i=1}^{N}\left(\frac{fL}{d_e^5}\right)_i$

The flow resistance of the blowing well is written as $\left(\frac{fL}{d_e^5}\right)_b$ and the task is to determine the flow resistance of the relief well, $\left(\frac{fL}{d_e^5}\right)_r$ to find an expression for the relief wells diameter. The well should be designed so that the frictional pressure loss in the relief well is less or equal to the maximum allowable frictional pressure loss:

$$\Delta P_{fr} \le \Delta P_{fr_{max}} \text{ or } \frac{\Delta P_{fr}}{\Delta P_{fb}} \le \frac{P_{an-max}-P_{frac}+\Delta P_{hyd}}{P_s-\Delta P_{hyd}} \tag{2.8}$$

Where: P_{an-max} - the maximal injection pressure in relief well annulus, P_{frac} - fracturing pressure of the formation, ΔP_{hyd} - component of BHP due to hydrostatic weight of fluid and P_s - static formation pressure. All pressures are given in $[psig]$.

By setting in expressions for each parameter in the equation above (2.8), the equivalent diameter of the relief well can be calculated with:

$$d_{e_r} = \left[\left(\frac{d_e^5}{fL}\right)_b \frac{\Delta P_{fb}}{\Delta P_{fr}}\frac{1}{k^2} f_r L_r\right]^{1/5} \tag{2.9}$$

Where:
ΔP_{fb} is the frictional pressure loss in the blowing well: $\Delta P_{fb} = P_S - \Delta P_{hyd}$, and ΔP_{fr} is the frictional pressure loss in the relief well: $\Delta P_{fr} = P_{an} - (P_{frac} - \Delta P_{hyd})$

- Hydraulic horsepower (HHP):

Sufficient HHP is required to ensure that the pumps deliver enough kill fluid down the relief well to dynamically kill the blowing well. In HHP calculations leak-off should also be taken into consideration.

The hydraulic horsepower can be calculated with:

$$HHP = \frac{q_r\, P_{an-max}}{40{,}81} \tag{2.10}$$

Here: $q_r\ [bpm]$ is the injection rate down the relief well, and P_{an-max} is the maximum injection pressure down the relief well given by $P_{an-max} = \left(\Delta P_{f_b} + \Delta P_{f_r} + \Delta P_{f_c}\right)_{max}$, where $\Delta P_{f_b}, \Delta P_{f_r}, \Delta P_{f_c}$ are the frictional pressure losses in the blowing well, the relief well and in the communication channel between the wells, respectively.

- Maximum allowable BHP to prevent the drill pipe from being ejected from a vertical, blowout hole:

During drilling, the drill string is exposed to forces tending to eject it from the well. The ejecting force is composed of a hydraulic force and frictional drag acting on various cross sections of the drill string.

The hydraulic force is given by $F_H[pounds] = \frac{\pi}{4}\, d_1^2\, P_{BH}$, where d_1 - OD of drill pipe, (d_0 - ID of casing or open hole) and P_{BH} - BHP [psig]. The total drag is calculated by determining the frictional pressure drop and multiplying this over the cross sectional area of the flow: Total drag = $\Delta P_f\, A_{an}$. *"The total frictional drag is applied to both the inside surface of the casing and the outside surface of the drill string. The ratio (R) of the total frictional drag that applies to the inner string is determined by the ratio of the shear stresses [14]"*: $R = \frac{1}{2\, ln\left(\frac{d_0}{d_1}\right)} - \frac{d_1^2}{(d_0^2 - d_1^2)}$

If the ejection force is greater than the effective weight of the drill string (in air), $W_s[pounds]$, the drill string may be ejected from the well leaving an increased flow area for the kill fluid and a lower frictional pressure. To kill the well several relief wells may then be required to obtain a sufficient flow rate creating enough frictional pressure.

The force balance between the ejecting forces and the drill string weight can be written as:

$$\frac{\pi}{4}d_1^2 P_{BH} + \frac{\pi}{4}(d_0^2 - d_1^2)R\Delta P_f \leq W_s \tag{2.11}$$

ΔP_f can be calculated from the equation $\Delta P_f[psig] = P_{BH} - \Delta P_{hyd}$ leading to the following expression for the maximum allowable BHP to keep the drill string from being ejected:

$$P_{BH\,max}\,[psig] = \frac{W_s + A_{an}R\,\Delta P_{hyd}}{A_{dp} + A_{an}R} \tag{2.12}$$

Here: A_{an} and A_{dp} are the areas $[inches^2]$ of the annulus and drill pipe, R is the ratio of total frictional drag on the drill string, and W_s is the weight $[pounds]$ of the drill string in air.

The author refers to the appendices in reference [14] for more detailed description and calculations-steps of the design parameters.

Two-phase flow is generally difficult to describe. The design parameters presented above are simple and neglect the influence of flow regimes, well geometries, multiple phases flowing etc. This raises a lot of uncertainty for the dynamic kill operation. The consequences of neglecting important parameters may cause under or over prediction of the dynamic kill; loss of the only chance to kill the well and waste of resources and time may be the results. All parameters, except the HHP parameter, are therefore replaced with more complex expressions when performing simulations with two-phase dynamic kill models. This is to achieve realistic results to perform a safe well planned dynamic kill operation.

2.2.2. Optimal Flow Rate – An Alternative Approach

In the following an alternative approach on how to find the optimal flow rate will be given based on reference [27]. The calculations in this section are valid but yield conservative kill-rate predictions. The method takes into account whether the flow is turbulent or laminar through the Moody friction factor, and also accounts for the kill fluids leak-off. However, important parameters like slip between the flowing phases and flow regimes are neglected.

The method finds the minimum kill fluid injection rate by first determining the kill fluid rate required to produce a pressure equivalent to the reservoir pressure, p_R.

The flowing pressure, p_{wf} $[psi]$, for a blowing well can be described by the energy equation:

$$p_{wf}[psi] = p_{wh} + \frac{g}{g_c}\rho h + \frac{\rho(v_{wh}^2 - v^2)}{2g_c} + \frac{\rho f_M v^2 L}{2g_c d} \qquad (2.13)$$

Where:

- p_{wh} $[psi]$ is the outlet pressure at wellhead.
- $\frac{g}{g_c}\rho h$ is the hydrostatic pressure of the wellbore fluids: ρ $[lbm/gal]$ is the density of the blowing fluid(s), h $[ft.]$ is the vertical height, g $[ft/s]$ is the acceleration of gravity and g_c is the gravitational system conversion constant (32,17 $(lbm - ft)/(lbf - s)$).
- $\frac{\rho(v_{wh}^2 - v^2)}{2g_c}$ is the kinetic energy term: v $[ft/s]$ is the fluid velocity in the well and v_{wh} $[ft/s]$ is the fluid velocity at wellhead.
- $\frac{\rho f_M v^2 L}{2g_c d}$ is the frictional loss term where L $[ft]$ is the length, and d $[inches]$ is the hydraulic diameter. The friction factor, f_M, is the Moody friction factor given for a Newtonian fluid[3] in turbulent flow:

$$f_M = \left[1.14 - 2\log\left(\frac{\epsilon}{d} + \frac{21.25}{N_{Re}^{0.9}}\right)\right]^{-2}$$

Here ϵ $[inches]$ is the pipe roughness and N_{Re} is the Reynolds's number determined by using $N_{Re} = 928 * d\ [inches] * v \left[\frac{ft}{s}\right] * \rho \left[\frac{lbm}{gal}\right] / \mu[cp]$. For flow in round pipe the hydraulic diameter $d = d_c$ (the flow conduit diameter), and in annular flow: $d = d_h - d_o$, where d_h and d_o are the hole and pipe diameters, respectively. $\mu[cp]$ is the fluid viscosity.

Equation (2.13) only applies to the flowing pressure at formation depth if the wellbore fluid is consistent and incompressible.

The reservoir section is the source of the blowout, and delivers formation fluids into the wellbore with a certain rate. To describe the deliverability of the reservoir a well inflow performance relationship, IPR curve, is determined. The IPR curve provides the flow rates at any back pressure and can for a gas well in a pseudo-steady state be expressed as:

[3] Fluids with constant viscosity at all shear rates at constant temperatures and pressures.

$$q_{gsc}[MMcf/D] = J\left(p_R^2 - p_{wf}^2\right)^n \qquad (2.14)$$

Here q_{gsc} is the gas flow rate at standard conditions, and J is the productivity index. "*The exponent n accounts for turbulence, and theoretically ranges from* $0{,}5 - 1{,}0$ [27]"

Equation (2.13) will for given liquid and blowout rates describe system-intake curves. By plotting these curves against the IPR curve (eq. 2.14), the minimum liquid flow rate needed to be injected for a successful dynamic kill can be found graphically: Figure 2-3 shows an example of an IPR curve plotted against system-intake curves for several water injection rates. "*A successful dynamic kill requires that the system-intake curve remains above or tangent the IPR at all blowout rates* [27]." From the case illustrated in Figure 2-3, the minimum kill rate lies between 80 - 85 bbl/min.

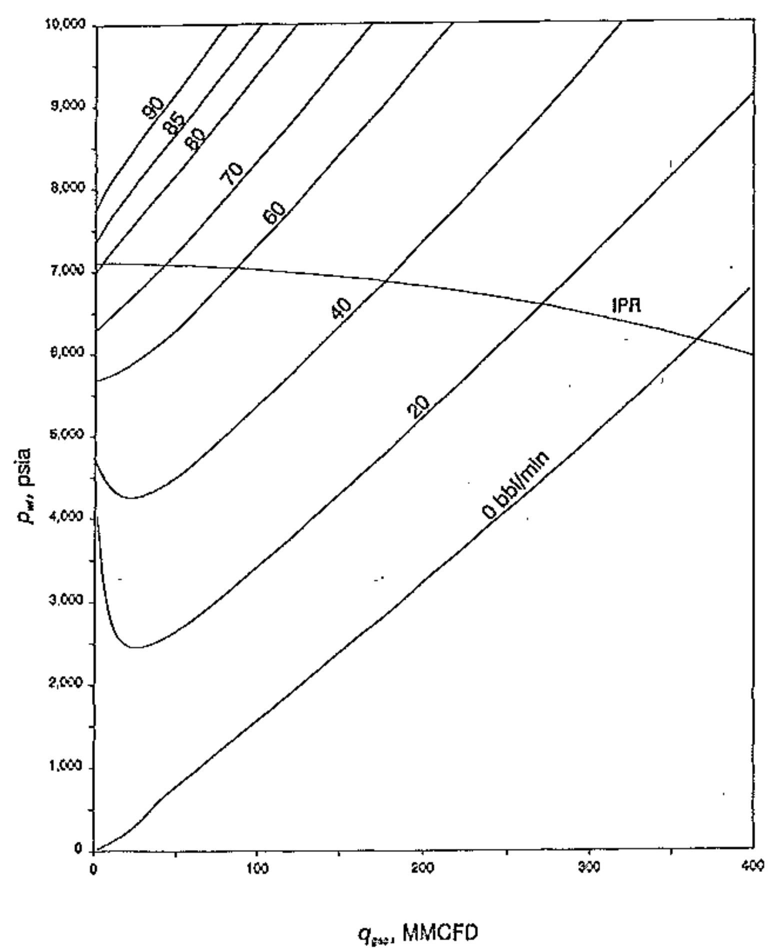


Figure 2-3 - IPR and System-Intake Curves for Different Injection Rates for the Arun Blowout Well C-II-2 and Relief Well C-II-8 [27]

The minimum dynamic kill rate can also be derived from equation (2.13) by setting p_{wh} equal to p_R , and ignoring any changes in kinetic energy:

For pipe flow: $$(q_l)_{min}[bbl/min] = 0.135 \left[\frac{(\rho_l - \rho_g) d_c^5 h_l}{(\rho_l + \rho_g) f_M L}\right]^{0,5} \tag{2.15}$$

For annular flow: $$(q_l)_{min}[bbl/min] = 0.135 \left[\frac{(\rho_l - \rho_g)\left(d_h^2 - d_o^2\right)^2 (d_h - d_o) h_l}{(\rho_l + \rho_g) f_M L}\right]^{0,5} \tag{2.16}$$

Since the friction factor depends on the Reynolds's number iteration will be necessary to solve for the minimum liquid rate.

When performing a dynamic kill on a gas blowout there is a possibility of getting counter current flow; low gas velocity makes kill fluid droplets fall and accumulate below the injection point in the wellbore. If any rock is exposed to the injected fluids the liquid rate in the blowing wellbore will not be the same as the injection rate. Fluid leak-off to the rock matrix must then be estimated and included in the minimum dynamic kill rate:

$$q_{kr}[bbl/min] = \frac{q_l}{E_{lo}} \tag{2.17}$$

Where: q_l $[bbl/min]$ is the liquid flow rate in the blowing well, and E_{lo} is the fluids leak-off efficiency.

3 Dynamic Modeling

3.1 Steady-State and Transient Two-Phase Models

This section is inspired by reference [19].

Multiphase flow models are widely used in the petroleum industry today and simulation of pipelines, blowouts and underbalanced drilling operations are only a few examples of were the models are used. The multiphase models are divided into two types: Steady-state models; where the pressures and flow rates is independent of time, and transient models; where the flow rate changes with time. The models are used to describe two-phase flow by using a two-fluid model consisting conservation equations for each of the phases with respect to mass, momentum and energy:

Steady-state model:

1. $(AH_i\rho_iU_i)_{in} = (AH_i\rho_iU_i)_{out} + \psi$
2. $(AH_i\rho_iU_i^2)_{in} = (AH_i\rho_iU_i^2)_{out} + \sum F$
3. $\left(A\rho_gU_g\left(h_g + \frac{U_g^2}{2} + gy\right)\right)_{in} + \left(A\rho_lU_l\left(h_l + \frac{U_l^2}{2} + gy\right)\right)_{in} = \left(A\rho_gU_g\left(h_g + \frac{U_g^2}{2} + gy\right)\right)_{out} + \left(A\rho_lU_l\left(h_l + \frac{U_l^2}{2} + gy\right)\right)_{out} + Q$

Transient model:

1. $\frac{d}{dt}(VH_i\rho_i) = (AH_i\rho_iU_i)_{in} - (AH_i\rho_iU_i)_{out} + \psi$
2. $\frac{d}{dt}(VH_i\rho_i) = (AH_i\rho_iU_i^2)_{in} - (AH_i\rho_iU_i^2)_{out} + \sum F$
3. $\frac{d}{dt}\left[V\rho_g\left(e_g + \frac{U_g^2}{2} + gy\right) + V\rho_l\left(e_l + \frac{U_l^2}{2} + gy\right)\right] = \left(A\rho_gU_g\left(h_g + \frac{U_g^2}{2} + gy\right)\right)_{in} + \left(A\rho_lU_l\left(h_l + \frac{U_l^2}{2} + gy\right)\right)_{in} - \left(A\rho_gU_g\left(h_g + \frac{U_g^2}{2} + gy\right)\right)_{out} - \left(A\rho_lU_l\left(h_l + \frac{U_l^2}{2} + gy\right)\right)_{out} - Q$

Where:

A – Pipe cross sectional area $[m^2]$

$e_{g/l}$ – Internal energy of gas and liquid $[Joules/kg]$

$\sum F$– Sum of pressure, gravitational and friction forces acting on phase i $[N]$

g – Acceleration due to gravity $[m/s^2]$

H_i – hold up of phase i

Q – Heat loss $[Watt]$ across the pipe wall to the surroundings
V – Pipe section volume $[m^3]$
$U_{g/l}$ – Velocity of gas/liquid $[m/s]$
$\rho_{g/l}$ – Density of gas/liquid $[kg/m^3]$
ρ_i – Density of phase i $[kg/m^3]$
y – Vertical position of pipeline $[m]$
Ψ – Rate of mass exchange due to condensation/evaporation $[kg/s]$

To solve for the parameters $\sum \mathrm{F}$, Q, Ψ, and simulate the flow conditions the models require the use of closure laws. This will be elaborated in section 3.3. The combination of conservation equations and closure laws in a steady-state or transient model results in complex equations e.g. partial differential equations which are difficult to solve analytically. The models are therefore solved numerically and implemented in a software tool.

During simulation of a dynamic kill, the steady-state and the transient models operate differently. The steady-state model uses an iterative method when predicting the flow rate and pressure. The model starts by estimating a pressure in the wellbore based on exit conditions, and uses this as a starting point to calculate reservoir parameters and pressure profiles for the wellbore. The steady-state equations are then used to back step through the input geometry from the exit to the reservoir, the result is a pressure profile based on the exit conditions. The pressure profiles are compared to the reservoirs IPR curve; if the IPR curve match the flowing bottom hole pressure a solution is reach, if there is no match exit conditions are adjusted and calculations are repeated until a convergence is reached. [33]

The transient two-phase model uses a slightly different approach than the steady-state model: "*The model calculates the changes in BHP at successive intervals as fluid is pumped in* [33]" taking into account time-depended changes in friction and reservoir parameters. An IPR curve is used to determine the volume of formation fluid flowing into the wellbore as kill fluid is pumped in. The model steps through the kill using different kill fluid volumes and pumping rates displaying its results and findings through plotted curves. The curves give an indication on which kill fluid and kill rate that should be used to obtain a successful dynamic kill. [33]

In certain situations for example in pipeline start-up or shutdown, it is easy to see that the situation is time-dependent and hence a transient flow model should be used. But sometimes choosing between the two models can be challenging. There are situations where multiphase flow in pipelines is steady but where the flow behavior within the pipeline has a strong transient character e.g. terrain slugging. Therefore it is important to see the models as complementary rather than competitive models.

3.2 The Drift-Flux Model

This section is based on references [18] and [20].

Since the conservation equations are fairly complex (ref. Section 3.1) the two-phase models are often simplified. The simplification can be done by adding the momentum conservation equations into a mixture momentum equation, and neglecting the energy equation due to the assumption of a fixed temperature gradient determined as a function of the geothermal gradient. The result is a drift-flux model which focuses on the movement of the two-phase fluid as a whole rather than as two-phases separately.

For an isothermal[4] two-phase flow of gas and liquid the drift-flux model can be expressed as [20]:

$$\begin{aligned}
&1)\quad \frac{\partial}{\partial t}\left(A\alpha_g\rho_g\right) + \frac{\partial}{\partial x}\left(A\alpha_g\rho_g v_g\right) = A\Gamma_g \\
&2)\quad \frac{\partial}{\partial t}\left(A\alpha_l\rho_l\right) + \frac{\partial}{\partial x}\left(A\alpha_g\rho_g v_g\right) = -\,A\Gamma_l \\
&3)\quad \frac{\partial}{\partial t}\left(A\left(\alpha_l\rho_l v_l + \alpha_g\rho_g v_g\right)\right) + \frac{\partial}{\partial x}\left(A\left(\alpha_l\rho_l v_l^2 + \alpha_g\rho_g v_g^2\right)\right) + A\frac{\partial}{\partial z}p = -\,Aq
\end{aligned} \tag{3.1}$$

Where:

α_l, α_g – Volume fractions of liquid and gas

ρ_l, ρ_g – Densities of liquid and gas

v_l, v_g – Velocities of liquid and gas

Γ_l, Γ_g – Mass exchange between the two phases, liquid and gas

p – Pressure

A – Flow area

$q = K + \rho_{mix}g$ – Represents the external forces acting on the fluids, where K is the friction pressure loss, and $\rho_{mix}g = (\rho_g\alpha_g + \rho_l\alpha_l) * g$ is the gravitational forces.

The model presented in equation (3.1) is a transient drift-flux model. "*The two first equations represent the mass transport of gas and liquid, and the third represents the total momentum balance, stating that the total pressure gradient depends on friction, gravity and acceleration*

[4] For isothermal flow the temperature remains constant: ΔT = 0

[18].” By using a mixture momentum equation the difficult terms related to phase interactions cancel out, and is replaced with an empirical slip equation giving a relation between the phase velocities. “*The source terms related to mass transfer, friction and gravity are still present in the model.* [20]”

The drift-flux model (3.1) can be further simplified by neglecting flow area changes [20]:

$$\frac{\partial}{\partial r}\boldsymbol{w} + \frac{\partial}{\partial x}F(\boldsymbol{w}) = G(\boldsymbol{w}) \tag{3.2}$$

Where:

$$\boldsymbol{w} = \begin{pmatrix} \alpha_l\rho_l \\ \alpha_g\rho_g \\ \alpha_l\rho_l v_l + \alpha_g\rho_g v_g \end{pmatrix}, F(\boldsymbol{w}) = \begin{pmatrix} \alpha_l\rho_l v_l \\ \alpha_g\rho_g v_g \\ \alpha_l\rho_l v_l^2 + \alpha_g\rho_g v_g^2 + p \end{pmatrix}, G(\boldsymbol{w}) = \begin{pmatrix} 0 \\ 0 \\ -q \end{pmatrix} \tag{3.3}$$

The expression in (3.3) can also be written as:

$$\partial t \begin{pmatrix} w_1 \\ w_2 \\ w_3 \end{pmatrix} + \partial x \begin{pmatrix} v_l w_1 \\ v_g w_2 \\ v_l^2 w_1 + v_g^2 w_2 + p(w_{1,} w_2) \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ -q \end{pmatrix} \tag{3.4}$$

Where $w_1 = \alpha_l\rho_l, w_2 = \alpha_g\rho_g$ and $w_3 = \alpha_l\rho_l v_l + \alpha_g\rho_g v_g$. Pressure and phase volume fractions are functions of the conservative variables w_1 and w_2.

3.3 Closure Laws

The drift-flux model is an approximation of the reality and the accuracy of the model depends heavily on using proper closure laws for flow pattern description, phase interaction, pressure losses, densities, slip relations between the phases etc. The common way to determine the closure laws is by performing experiments, but some laws can also be determined theoretically for example closure laws for various equations for the state of gas.

Examples of closure laws are [21]:

- Friction models: Friction depends on several parameters; geometry, velocity, viscosity, pressure, temperature and density. During simulation it is therefore important to ensure that the friction model used covers all the parameters necessary to obtain a realistic scenario.

- Density models: The density of a two-phase mixture is easy to calculate provided that the fluid fractions, $\varepsilon_{l,g}$, are known. The mixture density of two flowing fluids can be written as: $\rho_m = \rho_l \varepsilon_l + \rho_g \varepsilon_g$

- Viscosity models: Viscosity for a two-phase flow is not a well-defined quantity because it strongly depends on dynamic processes and flow regimes and therefore many models[5] exist. In the simulation in Chapter 5 the Dukler model for the mixture viscosity is used: $\mu_{mix} = \varepsilon_g \mu_g + (1 - \varepsilon_g)\mu_l$

- Slip models: Slip is the phenomenon in multiphase flow when one phase flows faster than the other phase and slips past it [4]. For two phases (gas and liquid) the slip velocities and ratio are defined by $u_S = u_G - u_L$ and $S = \frac{u_G}{u_L}$, respectively.

  Generally for a mechanistic model, the slip relation has the following form [22]:

  $$v_G = C_0(\alpha_G v_G + \alpha_L v_L) + v_0 \tag{3.5}$$

  Where the parameters $v_G, v_L \left[\frac{m}{s}\right]$ are the in-situ velocities of gas and liquid, C_0 is the distribution parameter and v_0 $[m/s]$ is the drift velocity. C_0 and v_0 depend on the flow regime.

In the following simple closure models for slip, density and friction will be reviewed for a two-phase model based on [20]. A simple drift-flux model for gas and liquid flow is considered. No mass transfer between the two phases is assumed.

$$\Gamma_l = \Gamma_g = 0 \tag{3.6}$$

For the gas phase, the following slip law applies:

$$v_g = K v_{mix} + S \tag{3.7}$$

[5] Reference is given to [21] for description of the different models.

Where $v_{mix} = \alpha_l v_l + \alpha_g v_g$ is the average mixture velocity, and K and S are flow dependent parameters. When K = 1.0 and S = 0, no-slip conditions between the phases exists.

The closure laws for the densities of gas and liquid is based on the velocities of sound in each of the two phases:

For the gas phase an ideal gas law is assumed, $\rho_g = \frac{p}{\alpha_g^2}$, where $\alpha_g = 316\, m/s$ is the sound velocity in gas and $p\ [Pa]$ is the pressure.

While the liquid density is given by $\rho_L = \rho_{l,o} + \frac{p - p_{l,o}}{\alpha_l^2}$ (3.8)

Here $\alpha_l = 1000\, m/s$ is the sound velocity in liquid, and $\rho_{l,o}$, $p_{l,o}$ are constants assumed to be $1000\ kg/m^3$ and $1\ bar$, respectively.

In the two-phase flow both gravitational and frictional forces (viscous forces and forces between the wall and fluids) interact. The two forces are expressed by the source term:

$$q = F_w + F_g \quad (3.9)$$

Where:

$F_g = g(\alpha_l \rho_l + \alpha_g \rho_g) \sin\theta$, here g is the gravitational constant and θ is the inclination.

$$F_w = \frac{32 v_{mix} \mu_{mix}}{d^2} \quad (3.10)$$

Where: d is the pipe diameter and μ_{mix} is the mixed viscosity given by $\mu_{mix} = \alpha_l \mu_l + \alpha_g \mu_g$. The friction model (3.10) is quite simplified only valid for laminar flow in pipe.

The closure models presented here are very simplified. For more accurate models for gas and liquid flow, one has to take into account flow regimes and use better corresponding models for gas slippage and friction.

3.4 Flow Regimes and Transitions

The following section is based on references [21] and [22].

In two-phase flow different flow regimes can develop depending on time and space and the distributions of the phases present.

For vertical and horizontal flow one can distinguish between the flow regimes:

- Stratified flow: A continuous liquid stream flowing at the bottom of a horizontal pipe, with a continuous stream of gas flowing over.
- Slug flow: A continuous liquid stream flowing with low velocity, punctuated by gas bubbles.
- Dispersed bubble flow: Liquid flow flowing with intermediate velocity containing small dissolved gas bubbles.
- Annular flow: A thin liquid film at the pipe wall, and a gas stream containing liquid droplets flowing in the middle of the pipe with a high velocity.

For vertical flow, churn flow can be seen on as an intermittent flow regime between slug and dispersed bubble flow; the velocity of liquid and gas is increasing and larger gas bubbles is starting to break and deform. Likewise for horizontal flow the flow regimes stratified wavy and elongated bubble flow are intermittent flow regimes for stratified flow and slug flow.

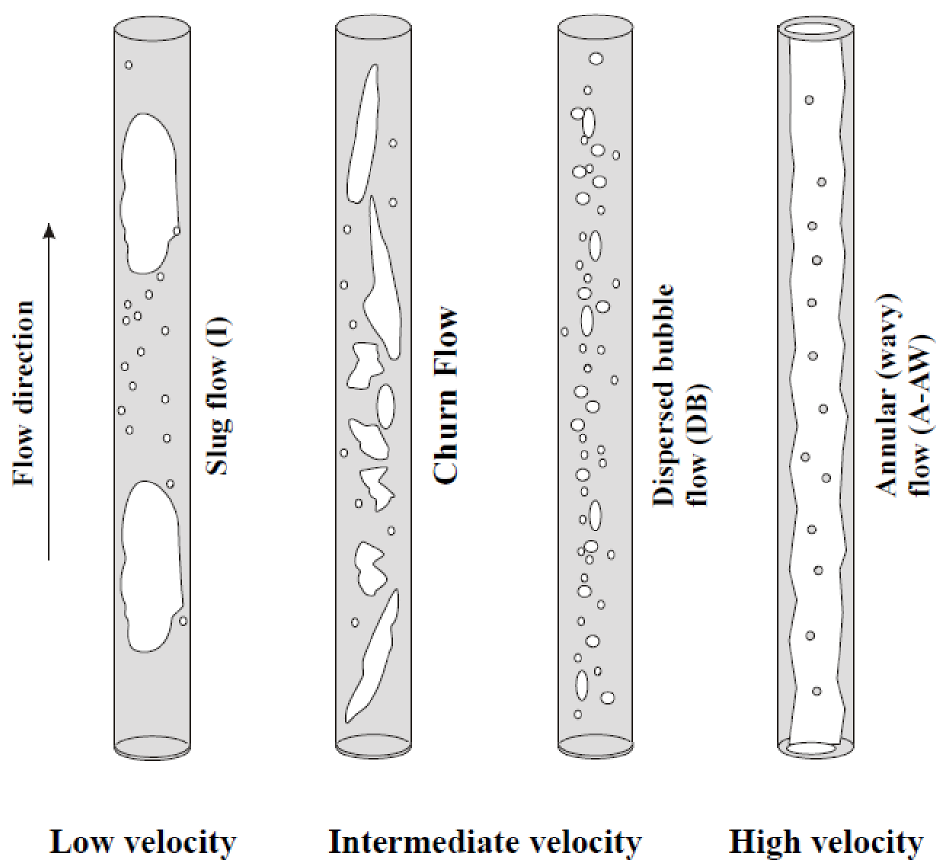


Figure 3-1 - Vertical Flow Regimes [21]

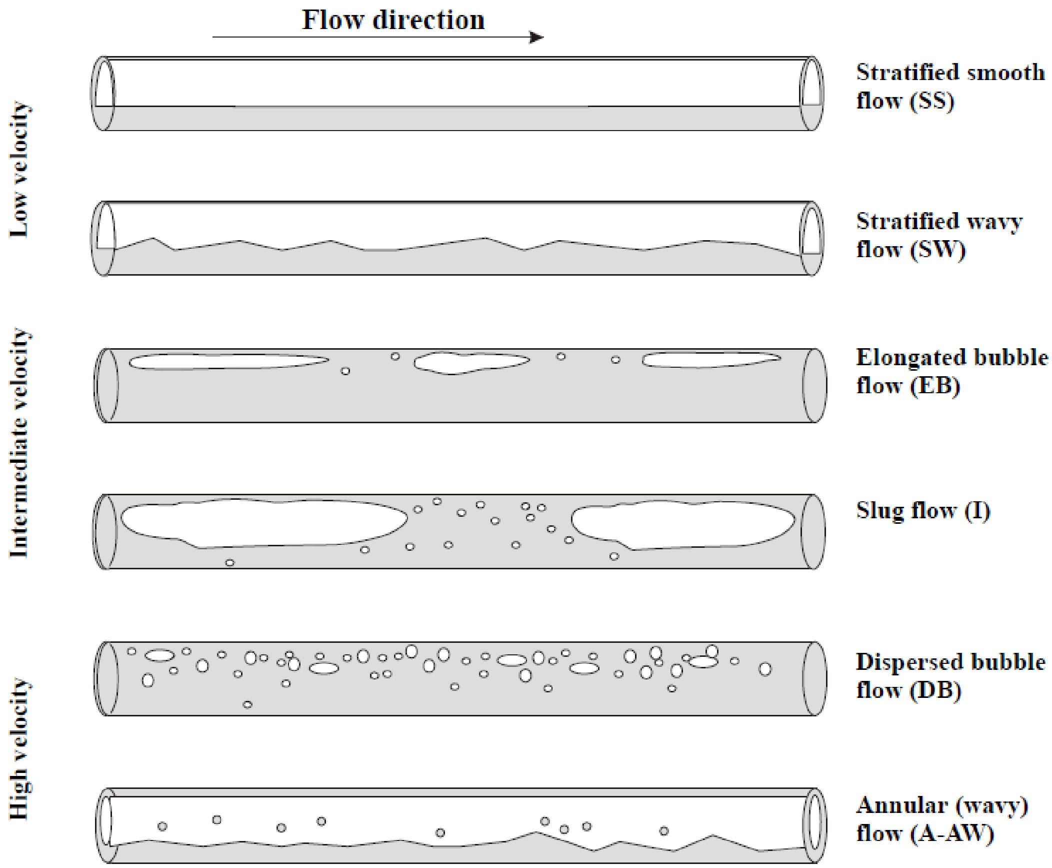


Figure 3-2 - Horizontal Flow Regimes [21]

In a mechanistic model for two-phase flow it is important to predict the flow regimes and the transitions to be able to describe the two-phase flow as accurate as possible. In the study [28] of Lage and Time a mechanistic model for upward two-phase flow in vertical and concentric annuli was presented. The model was compared to small and full-scale experimental tests and showed good agreement between model predictions and experimental data. In the following this model will be used to show how transition boarders between flow regimes can be modeled for a two-phase flow. For further details regarding Lage and Times mechanistic model reference is given to [28].

A two-phase flow of gas and liquid is assumed flowing in vertical concentric annuli with channel width larger than the capillary constant. The following geometrical parameters then apply:

$D_h[m] = D_2 - D_1$ and $D_{ep}[m] = D_2 + D_1$

Here D_h and D_{ep} are the hydraulic and equi-periphery diameters, respectively. D_1 is the outside diameter of the inner pipe and D_2 is the inside diameter of the outer pipe.

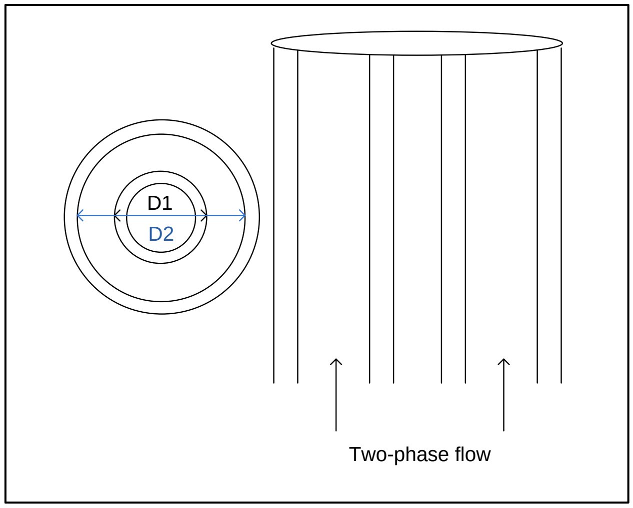


Figure 3-3 - Flow Geometry[6]

The geometrically limitation for the existence of bubble flow in annuli is given by:

$$D_{ep}[m] \geq 19{,}7 \left[\frac{(\rho_L - \rho_G)\sigma}{g\rho_L^2}\right]^{0,5} \tag{3.11}$$

Where: $\rho_L, \rho_G [lbm/gal]$ are the densities of liquid and gas, $\sigma\ [N/m]$ is the interfacial tension and $g\ [m/s^2]$ is the gravity acceleration.

If the equi-periphery diameter is larger than the expression on the right side, bubble flow will turn into slug flow represented by the following equation:

$$u_{Ls}[m/s] = \frac{1-\alpha}{\alpha} u_{Gs} - (1-\alpha)u_0 \tag{3.12}$$

Where $u_{LS}, u_{GS}\ [m/s]$ is the superficial velocities of liquid and gas, α is the gas volumetric fraction and $u_0 [m/s]$ is the slip velocity.

At low liquid rates gas bubbles are large and the rise velocity is less sensitive to the bubble size. This means that the following equation also applies for the rise velocity of a single bubble in an infinite medium:

[6] Figure 3-3 is inspired by Figure 3 in reference [28].

$$u_{0\infty}[m/s] = 1{,}53\left[\frac{(\rho_L-\rho_G)\sigma}{\rho_L^2}\right]^{0{,}25} \tag{3.13}$$

If a bubble is rising in a swarm of bubbles the equation above should be corrected by $u_0 = u_{0\infty}(1-\alpha)^n$, where $0{,}5 \leq n \leq 2{,}0$. In the rest of this section $n = 0{,}5$ is assumed. Equations (3.12, 3.13) describe the flow regime borderline marked A in Figure 3-4.

When the velocity of the liquid phase is increasing gas bubbles starts to break up. The borderline for breaking up larger bubbles into smaller can be described by the equation below indicating transition B in Figure 3-4:

$$2\left[\frac{0{,}4\sigma}{(\rho_L-\rho_G)g}\right]^{0{,}5}\left(\frac{\rho_L}{\sigma}\right)^{0{,}6}\left(\frac{2f_M}{D_h}\right)^{0{,}4}u_M^{1{,}2} = 0{,}725 + 4{,}15\left(\frac{u_{Gs}}{u_M}\right)^{0{,}5} \tag{3.14}$$

Where f_M is the fanning friction factor based on the mixture velocity, u_M $[m/s]$.

At very high gas fractions ($> 0{,}52$) bubble flow cannot exist, causing transition to slug. Equation (3.15) gives the borderline corresponding to transition C in Figure 3-4:

$$u_{Ls}[m/s] = \frac{1-\alpha}{\alpha}u_{Gs} = 0{,}92u_{Gs} \tag{3.15}$$

Annular flow occurs when large velocities of liquid and gas arise, and the flow is depended on having a minimum gas velocity for balancing the gravity and drag forces acting on the largest stable droplets:

$$u_{Gs}[m/s] = 3{,}1\left[\frac{(\rho_L-\rho_G)g\sigma}{\rho_G^2}\right]^{0{,}25} \tag{3.16}$$

The above equation (3.16) corresponds to transition D in Figure 3-4.

When suspecting annular flow other effects should also be considered to guarantee the occurrence; annular flow occurs if the calculated value for gas fraction is higher than one obtained by the following equation $\alpha = \frac{u_{Gs}}{u_{tb}} = \frac{u_{Gs}}{1{,}2u_M+0{,}35\sqrt{gD_{ep}}}$ (3.17), where subscript (tb) means Taylor bubble. Equation (3.17) is represented by transition E on Figure 3-4.

For each of these flow patterns, there will be separate models used to calculate different flow variable likes phase fractions, phase velocities and pressure gradients. The total pressure gradient will be given by $\left(\frac{dp}{dL}\right)_T = \left(\frac{dp}{dL}\right)_H + \left(\frac{dp}{dL}\right)_F + \left(\frac{dp}{dL}\right)_A$, modeling the pressure drop in the pipe.

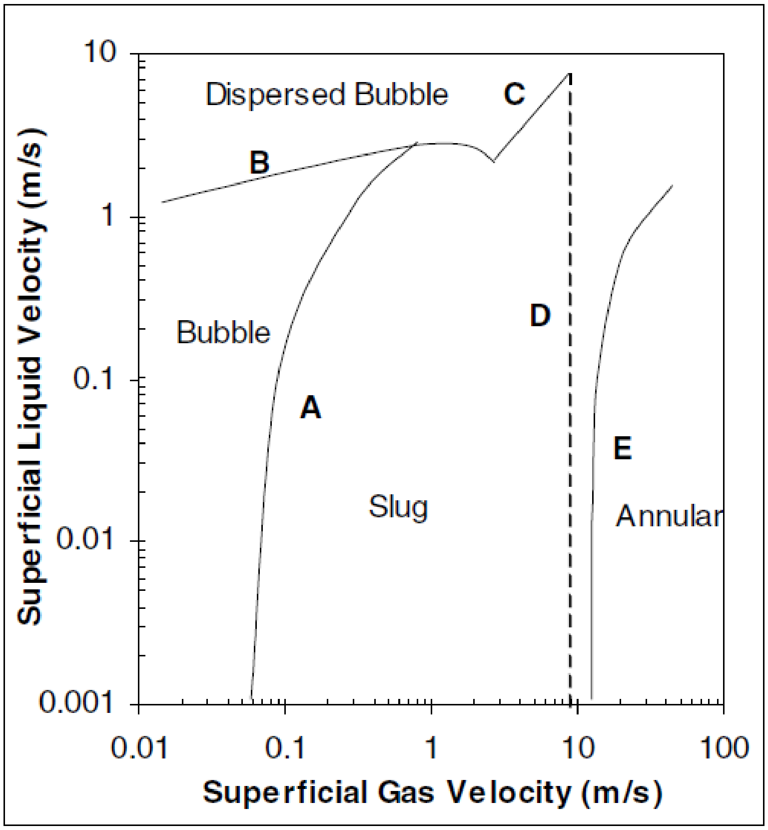


Figure 3-4 - Flow Pattern Map [28]

3.5 Discretization

To solve the drift-flux equations in (3.1) the well is divided into N boxes($i+1, \dots, i+n$) each with a hydrostatic pressure, a frictional pressure etc. depending on the well geometry and flow regime in the well. See Figure 3-5.

The conservation equations can be solved numerically using either an implicit or explicit numerical method. For an implicit solution the variables will be found based on "new" values. The result is a complex but fast solution; the time steps (Δt) can be larger and is only limited by the length of the boxes (Δx) divided by the maximum velocity of the flowing fluid (v). The explicit schemes solves the conservation equations by calculating new values based on old values; the time steps (Δt) is only limited by the length of the boxes (Δx), the velocity of the phase (v) and the speed of sound (c); $\Delta t < \frac{\Delta x}{v+c}$ [18]. This is the so-called CFL (Courant-Friedrichs-Lewy) - criterion.

As will be seen later, the numerical method used in chapter 5 will be an explicit method.

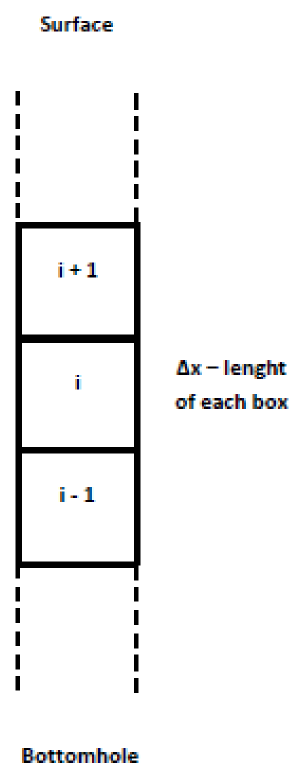


Figure 3-5 - Discretization

3.6 Shooting Technique

After the well has been discretized into several smaller boxes, one alternative can be to use a predictor-corrector shooting technique [29] to solve the equation system (3.1). Originally this method was developed to handle gas-kick situations but it can also be used for general two-phase simulations as described here [29].

When neglecting the acceleration terms related to the sonic waves, a no pressure wave model can be applied:

$$\frac{\partial}{\partial x}p = -q \qquad (3.18)$$

The shooting-technique starts by making a guess for the inlet pressure in the bottom of the well. A predictor-corrector algorithm is then used to solve the mass conservation equations (3.1) and the above equation (3.18). The solution from the algorithm gives the downstream flow variables in the first cell which is used as the upstream flow variables for the next cell, due to the assumption of continuity of pressure and mass over the cell boundary. By repeating this procedure for all cells in the well the outlet pressure at the top of the well is determined and compared to given physical boundaries. If there is no match the inlet pressure is iterated until the calculated outlet pressure has converged.

The predictor and corrector steps in the algorithm are as follows [29]:

- Equation (3.19) is used to calculate the upstream pressure in cell i:

  $$p^*_{i+1/2} = p^{n+1}_{i-1/2} - \Delta x q^n_i \tag{3.19}$$

  Average cell pressure is then given by:

  $$p^*_i = \frac{p^*_{i+1/2} + p^{n+1}_{i-1/2}}{2} \tag{3.20}$$

  The estimated pressure is then used to calculate the mass distribution.

- The next step is to guess the downstream mixture velocity $\left(v^*_{mix,\ i+\frac{1}{2}}\right)$ and find the new temporary mass in cell i:

  $$M^*_{k,i} = M^n_{k,i} + \Delta t f^{n+1}_{k,i-1/2} - \Delta t f^*_{k,i+\frac{1}{2}}, \quad k = l, g \tag{3.21}$$

  Where f represents mass fluxes between cells.

- The volume balance equation below is solved by iteration to determine the downstream mixture velocity:

  $$g\left(v^*_{mix,i+1/2}\right) \equiv V^*_i\left(v^*_{mix,i+1/2}\right) - A\Delta x = 0 \tag{3.22}$$

  Where: $V^*_i\left(v^*_{mix,i+1/2}\right) = \frac{M^*_{l,j}}{\rho_l(p^*)} + \frac{M^*_{g,i}}{\rho_g(p^*)}$

The corrector steps consist of repeating the presented steps above, equation (3.19) – (3.22), with updated flow variables until convergence of the flow variables is fulfilled.

A similar shooting approach can also be used for solving the steady-state equations given in section 3.1.

3.7 The AUSMV Scheme

The AUSMV (Advection Upstream Splitting Method) scheme is an alternative numerical method for analyzing two-phase flow. The scheme is easy to implement and previous work has shown that the scheme produces robust and accurate results for two-phase flow [20, 32].

The AUSMV scheme is used as basis for the numerical simulation in chapter 5 and a brief description of the scheme will be given in this section. References is given to [20, 31] for more details regarding the scheme.

"*The basis for the AUSMV scheme is the hybrid flux-vector splitting scheme (FVS)* [31]". After discretizing the well into smaller boxes, variables at a new time level (n+1) can be expressed as:

$$w_{i,j}^{n+1} = w_{i,j}^{n} - \frac{\Delta t}{\Delta x}\left(F_{j+1/2}^{AUSMV} - F_{J-1/2}^{AUSMV}\right) - \Delta t q_i^n \qquad (3.23)$$

The expressions for the fluxes, F, can be found in [20, 31] and are depended on the velocity of sound in the two-phase mixture.

The fluxes are treated explicit in time meaning that the values are based on "old" values (n) when calculations of the system are performed. This puts a restriction on which time steps used in the calculations. The allowed time step is limited by a CFL criterion:

$$\Delta t = CFL \frac{\Delta x}{max\langle \lambda_1 | \lambda_2 | \lambda_3 \rangle} \qquad (3.24)$$

Where λ_i is the eigenvalues.

To treat the information going in and out of the system, boundary conditions need to be established. At the inlet boundary the mass flow rates of gas and liquid are given, and by extrapolating the inlet pressure can be found:

$$P_{inlet} = p(1) + 0{,}5[p(1) - p(2)] \qquad (3.25)$$

Assuming a closed-in well ($V = V_g = V_l = 0$), the outlet pressure is found by extrapolation:

$$P_{outlet} = p(N) + 0{,}5[p(N) - P(N-1)] \qquad (3.26)$$

If an open-well is assumed the outlet pressure must be specified and fluxes extrapolated.

In wells and pipelines there may arise small or big flow area changes which must be taken into consideration when modeling the two-phase flow. By dividing the fluxes into pressure flux terms and convective terms the numerical scheme takes the following form:

$$w_{i,j}^{n+1} = w_{i,j}^{n} - \frac{\Delta t}{\Delta x}\left(A_R F_{m,j+1/2} - A_L F_{m,j-1/2} + A_{avg}\left(F_{p,j+1/2} - F_{p,j-1/2}\right)\right) - \Delta t q_i^n \quad (3.27)$$

Where $w_{i,j}$ is the mass conservative variable and momentum conservative variable for i = 1, 2 and 3, F_m represents the convective fluxes and F_p represents the pressure fluxes. A_R, A_L and A_{avg} are related to the flow area changes assumed in a cell. The average flow area change is defined as $A_{avg} = 0{,}5\ (A_L + A_R)$ and "*the subscripts R and L refer to the left and right hand side of the numerical cell* [31]". For further details reference is given to [31].

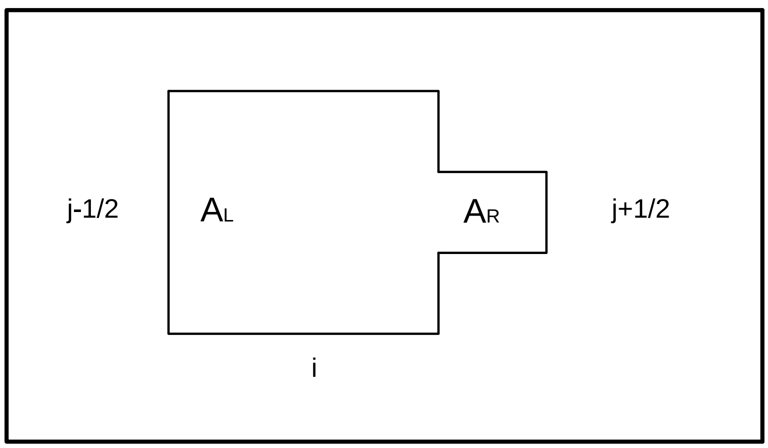


Figure 3-6 – Flow Area Changes

The fluxes can be found using special flux splitting formulas requiring the knowledge of the sound velocity in the fluid mixture. Using equation (3.27) for all the boxes gives the mass conservative and momentum conservative variables (w_1, w_2, w_3). Pressure and phase volume fractions can then be found from the mass conservative variables while phase velocities are found from the momentum conservative variable and the gas slip equation.

4 Software Models

Over the last decades several software models have been developed to assist in operations related to multiphase simulation in petroleum activities. Multiphase flow simulation has added a great value to the industry by giving the opportunity to build a virtual well or a pipe to analyze "what if" case scenarios and optimize on time and costs. The models also have a wide variety of use as support to decisions in areas like well planning, well control, pipeline transport and flow assurance making the models very economic feasible.

Multiphase models can be used to [23]:

- Ensure optimal use of gas lift and flow stability.
- Design pigging operations, underbalanced drilling operations etc.
- Optimize production.
- Detect kicks and simulate blowout scenarios.

For a dynamic kill model with focus on simulating blowouts the aim is to determine [6]:

- Down hole conditions such as pressures and flow rates.
- Requirements for the dynamic kill operation e.g. pump rate, mud volumes and kill fluid density.
- Better understanding of a possible blowout scenario; possibility of occurring, duration, environmental damage etc., and course of action.

<u>Gulf of Mexico Accident - The Macondo Blowout</u>

After the accident in the Gulf of Mexico, BP gathered an investigation team to analyze the events leading up to the blowout. As a part of the investigation the OLGA well-flow model [34], a transient multiphase well flow simulator, was used to simulate the events that took place before the accident. By matching OLGA simulation results to known real-time data, witness accounts, operational procedures, standards and policies, information on how the rig crew may have missed indications of an influx in the well and their response was collected. The OLGA well-flow model simulated the flow conditions and calculated the inflow rate and volume of hydrocarbons in the well. In addition, the model helped in detecting which barriers that had failed prior to the blowout.

The OLGA well-flow model enabled BP to go back in time and evaluate the accident. This was very helpful in learning from previously mistakes and preventing similar situations to occur again. Reference is given to [3] for detailed description of the findings from the BP investigation report.

The focus in this chapter will be on software models designed for simulation of blowouts and kill of blowouts. Other models of importance will also be discussed.

4.1 OLGA

OLGA is a dynamic two-fluid model used for simulating multi-phase flow systems. The software is commercialized by SPT Group [34] and is the marked-leading simulator for flow simulation of oil, gas and water in wells, pipelines and receiving process facilities.

A first version of OLGA was developed in 1983, but due to some difficulties with describing the stratified and annular flow regimes a new and extended version was later developed. The new development was a result of a research program performed by SINTEF [35] and the Institute for Energy Technology (IFE) [36] sponsored by several oil companies. [24]

OLGA`s difficulties with describing the stratified and annular flow regimes was explained by the neglect of a droplet field moving at approximately the gas velocity. This caused over prediction in pressure by 50% in vertical annular flow and over prediction in holdups in horizontal flow by a factor of two in extreme cases. To overcome the challenges and be able to describe the flow regimes and the corresponding transitions, the OLGA model was extended by introducing conservation equations for the gas phase, liquid phase at the wall and for the liquid droplets field moving in the gas phase. By using the Moody equation at the wall and an empirical correlation for the gas-liquid interfacial friction depending on the flow regime, the frictional forces could be calculated. The flow regime of the two-phase flow was selected based on a "minimum-slip" criterion: "*The model selects the flow regime which gives the lowest difference between the gas and liquid velocities. The minimum slip condition also corresponds to the regime which gives the lowest liquid holdup for a given pressure drop*" [19].

The OLGA model has been verified and compared with data from the SINTEF two-phase flow laboratory. In addition, it has also been compared against field data giving good results. This verification is an ongoing process to maintain the models at the forefront of the multiphase technology.

4.2 OLGA Well-Kill

This section is inspired by reference [16].

OLGA Well-Kill is a modified version of the original OLGA simulator and is a software program used for well control simulation. OLGA Well-Kill uses the core code from OLGA with added elements for drilling, production and well control operations. The software was developed through a research and development project assessing the experience gained from a blowout in 1989 in the Norwegian North Sea. Currently the software is provided to the industry by Add Well Flow AS - a part of the Add Energy Group [25].

The OLGA Well-Kill simulator is designed to simulate well-kill scenarios in contingency planning through calculations of required pump rates and hydraulic horsepower, kill volumes and operational time. The simulator can perform dynamic kill and top kill simulations as well as bullheading, momentum and shallow gas analysis.

Some of the advantages with the model are:

- Simulation with different well flow configurations (single well - multiple wells, vertical well – horizontal well).
- Full modeling of oil and gas properties.
- Use of various models for reservoir inflow.
- Modeling of pressure and temperature effects on mud properties.
- Sensitivity analysis of kill fluids (water, brine, mud) with different densities and viscosities.
- Graphical presentation of the results.

The model can also be used to find a relief well strategy before drilling begins and point out design changes making the operation safer and cheaper.

The development of the OLGA Well-Kill simulator included the implementation of several sub models in order to simulate the well flow during drilling, production and well control. The following are taken from reference [37], the latest, available and published information regarding these sub models that the author could find per June 2011. Hence, this may not be representative for the current modeling status of the software. For further details regarding these sub models the author refers to [37].

- Wellbore friction factor:
  To obtain correct wall and interfacial friction factors in the annulus (annular geometry) when fluids are flowing, the hydraulic diameter was used to account for the increased wetted surface area compared to the flow area.

  For a single pipe the hydraulic diameter becomes $d_h = \frac{4A}{S}$, where A is the flow area and S is the wetted perimeter.

- Controller system:
  "*The function of the system is to keep parameters within specified boundaries by controlling variables like pump rate and choke setting*," e.g. if the parameters defined for the controller system for the maximum pump rate is exceeded the pump rate is automatically reduced.

- Fluid properties:
  Before simulation starts tables with fluid properties; density, viscosity, surface tension, and compressibility, must be developed for the reservoir fluids (gas and liquid). "*A standard pressure volume temperature (PVT) - analysis package is used to generate these tables*" along with known correlations from experimental tests.

- Multiple phases:
  Several phases can occur in the well: Gas, oil, mud, water and solids. To reduce the number of phases to suit a two-phase model no-slip between two of the phases is assumed.

  The density of the mixed hydrocarbon (gas and oil) phase is then written as $\rho_{mix} = \alpha_g \, \rho_g + (1 - \alpha_g) \, \rho_o$ where α_g,$(1 - \alpha_g)$ is the volume fractions and ρ_g, ρ_o is the densities of gas and oil. Together with e.g. mud, the two-fluid system consists of a hydrocarbon mixture and a mud fluid. To reduce the errors that follow from averaging the phase's different options exists, "*one may be to use original gas and oil properties for the flow predictions after averaging [37]*".

- Non-Newtonian fluid flow:
  The core in the two-phase flow model assumes Newtonian flow, but models for non-Newtonian flow can also be chosen (flow dependent rheology).

- Reservoir inflow:
  Several different reservoir inflow models can be used depending on the type of reservoir simulated e.g. linear productivity index, quadratic productivity index, combined equations or tabulated IPR curves.

- Pumps and compressor models:
  Models for estimating pump power and volumes are added for the OLGA Well-Kill model to ensure simulation results fit for real kill operations.

- Flow path obstructions:
  Obstructions (packers, drill bit nozzles, valves etc.) may affect the total pressure drop in the well, but due to the low improvement on the results detailed modeling of these obstructions is often not necessary. Nevertheless, different sub-models (e.g. pressure loss models through chokes) have been developed and included in the OLGA Well-Kill model.

OLGA Well-Kill continues to be improved though research in multiphase behavior and through development work e.g. integration with other software packages [16].

4.3 OLGA ABC

OLGA ABC is based on the transient OLGA simulator and the Drillbench[7] application and is an advanced blowout control simulator. The model can simulate the actual blowout event for different scenarios, well types, drill string configurations and kill method (relief well), to calculate blowout rates, kill rates, required pumping duration and volumes as well as pressure loads in all well positions [38].

The software is used as operational support by verifying kill operations and contingency procedures. It is also used in evaluating environmental considerations by the determination of surface blowout rates.

OLGA ABC is currently provided by SPT Group [34].

[7] Transient two-phase flow simulator marketed by SPT Group [34]

4.4 BlowFlow

This section is based on references [39] and [40].

BlowFlow is a software tool and methodology developed for oil spill contingency by the International Research Institute of Stavanger (IRIS) [41]. The software tool uses traditional multiphase flow modeling combined with different killing methods to define blowout scenarios. The model takes into account uncertainty in different blowout input parameters and based on this it provides a probabilistic estimate of the duration of the blowout and which rates/blowout volumes that can occur. The software tool improves the communication within and between companies in blowout related issues by establishing a mutual documentation and calculation basis. This makes communication and decision-making across company boarders easier, hence meeting the recommendations stated in the Norwegian Oil Industry Association (OLF) report[8].

BlowFlow is a steady-state model with the ability to simulate the blowout from start to finish (dead well). The work process for the model is illustrated in Figure 4-1:

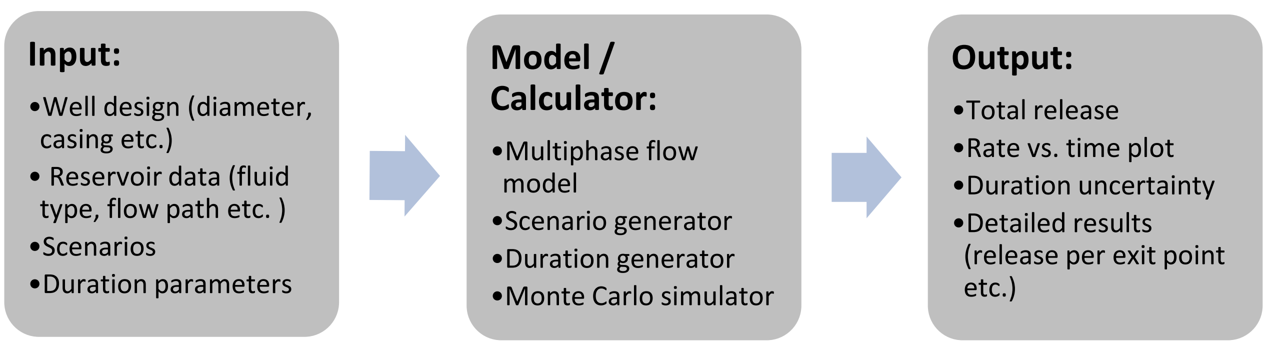


Figure 4-1 - BlowFlow Work Process[9]

In the Input-phase the well geometry and reservoir data are specified as well as blowout scenario and blowout stopping mechanism (capping, coning, bridging or relief well). It is also important to choose the correct distribution models to account for uncertainty in input data. Establishing the input parameters is the most time consuming part of the analyses as it comprises both certain and uncertain parameters. The certain parameters like well parameters are taken from the drilling program while the uncertain parameters e.g. reservoir pressures,

[8] Nilsen, T: "Retningslinjer for beregning av utblåsningsrater og – varighet til bruk ved analyse av miljørisiko", OLF/Statoil, Stavanger, Norway, 2004

[9] Figure 4-1 is based on Figure 3 in reference [39]

productivity index (PI) and gas oil ratio (GOR) need to be assessed by experts with the use of probability distributions as indicated in Figure 4-2.

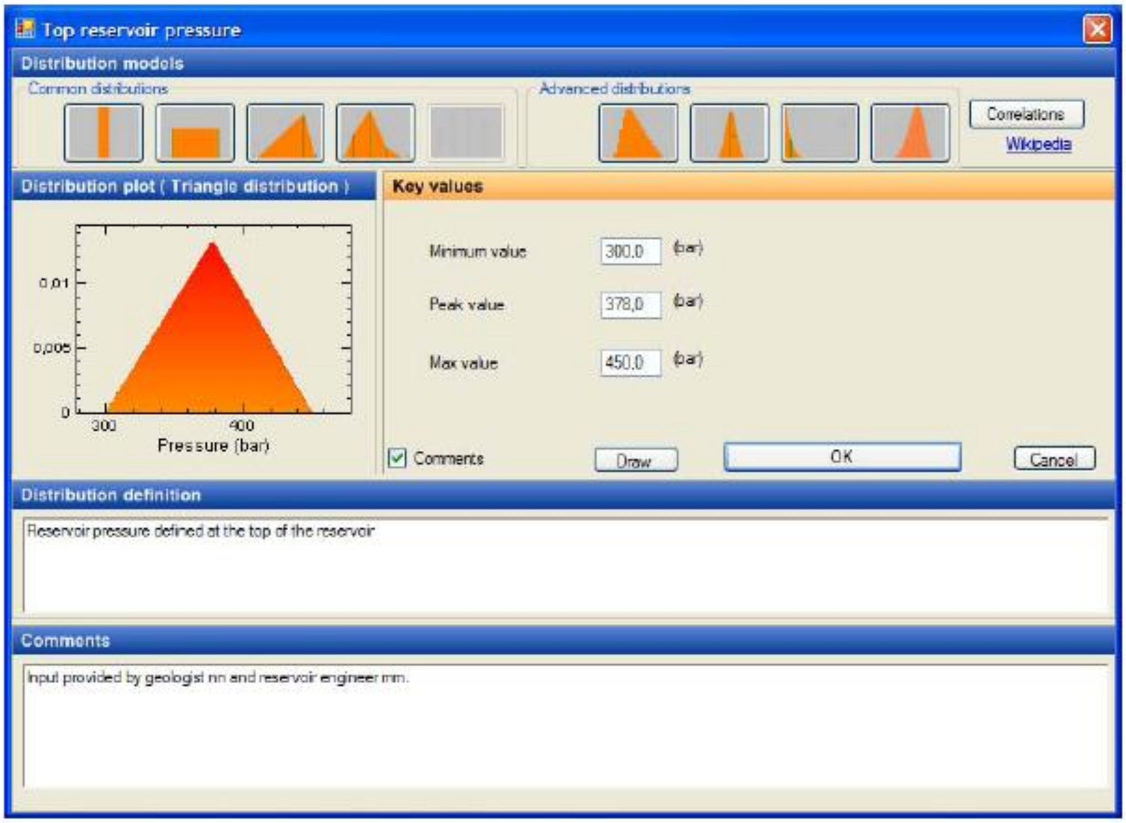


Figure 4-2 - Probability Distribution Used for Evaluating the Uncertainty Related to Reservoir Pressure [39]

After the input parameters are specified the next step is phase two – the calculation phase. The basis for this phase is Monte Carlo simulations. The simulation is performed by pre-defining a number of blowout scenarios and record what happens in each case. The results are presented in the last phase - the output phase. Here a summary of each of the blowout scenarios are given and expressed through probability distributions of rates, volumes and duration.

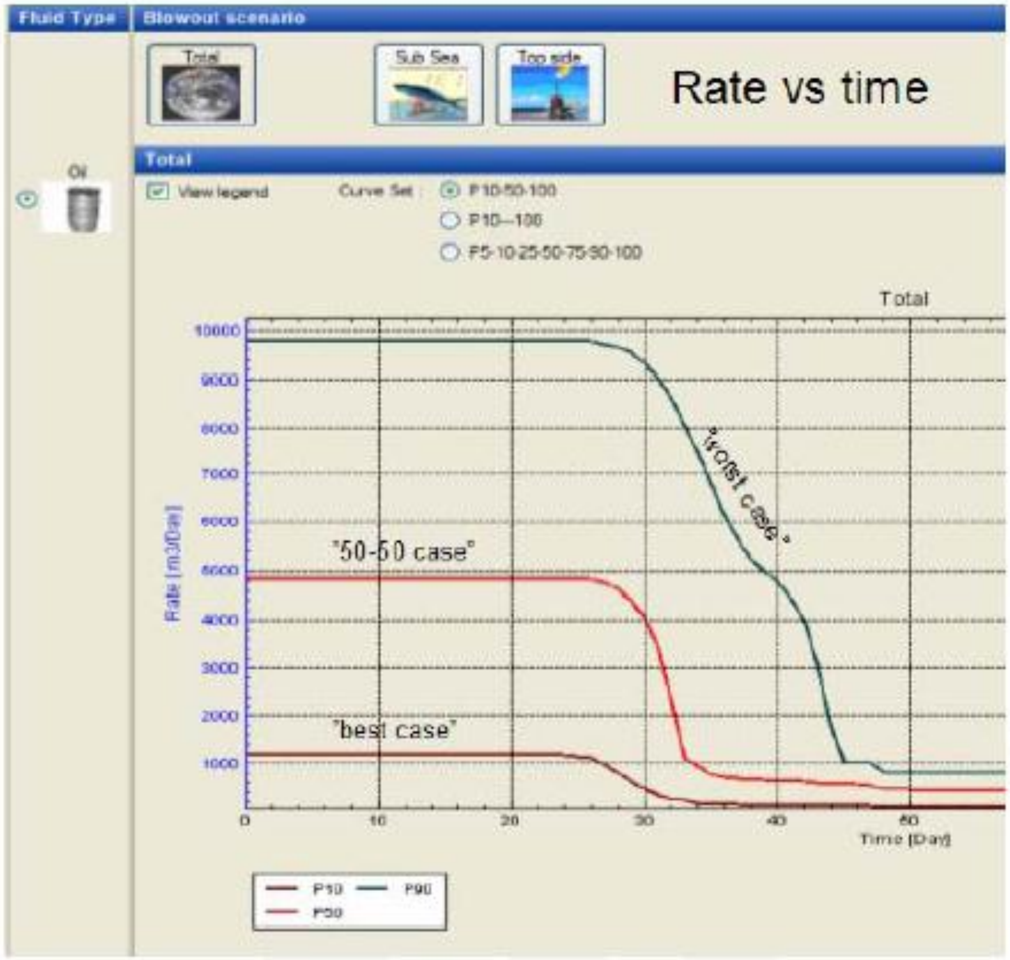


Figure 4-3 - Flow rate vs. Time [39]

Figure 4-3 shows a probabilistic representation of blowout rates vs. time for worst case, best case and most likely case for a blowout scenario. The accuracy of the results can be increased by increasing the number of Monte Carlo simulations [39].

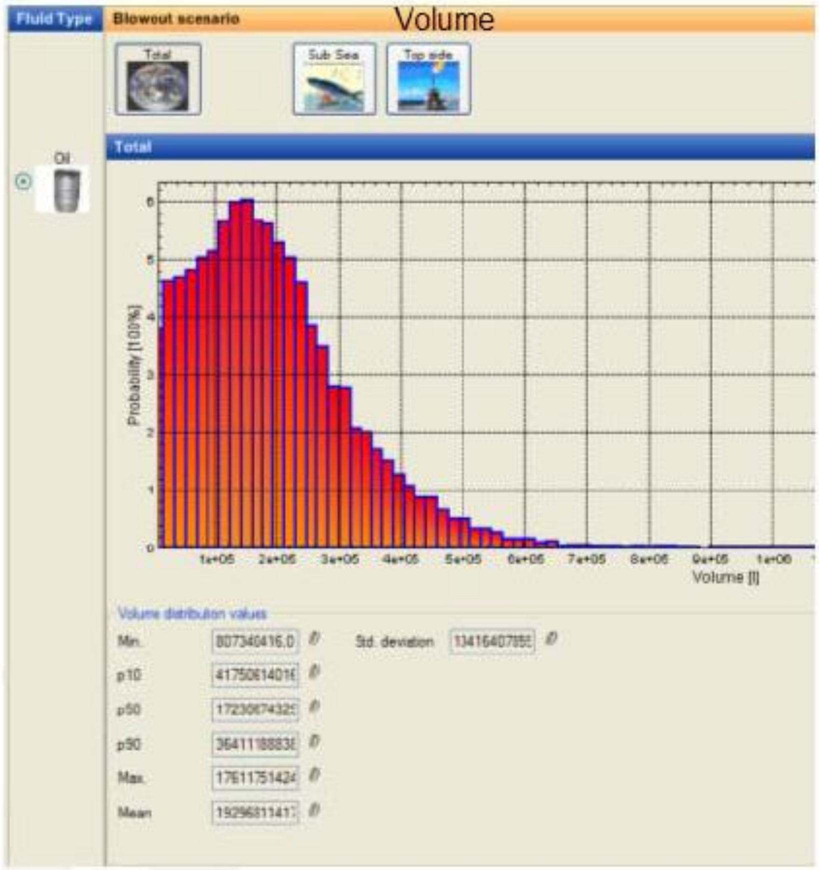


Figure 4-4 - Total Released Oil Volume [39]

The result from the BlowFlow model can be used as input in environmental risk analysis thus improving the decision making between involved parties as well as giving a good basis for evaluation of uncertainties related to blowout rates and volumes.

4.5 COMASim

COMASim (**C**herokee **O**ffshore, **M**MS, Texas **A**&M **Sim**ulator) is a simulator modeling the initial blowout conditions and the dynamic kill requirements either in a drill string or in a relief well – kill. The COMASim is written in Java code and can be accessed from the internet making it versatile for various platforms and operative systems. [42, 43]

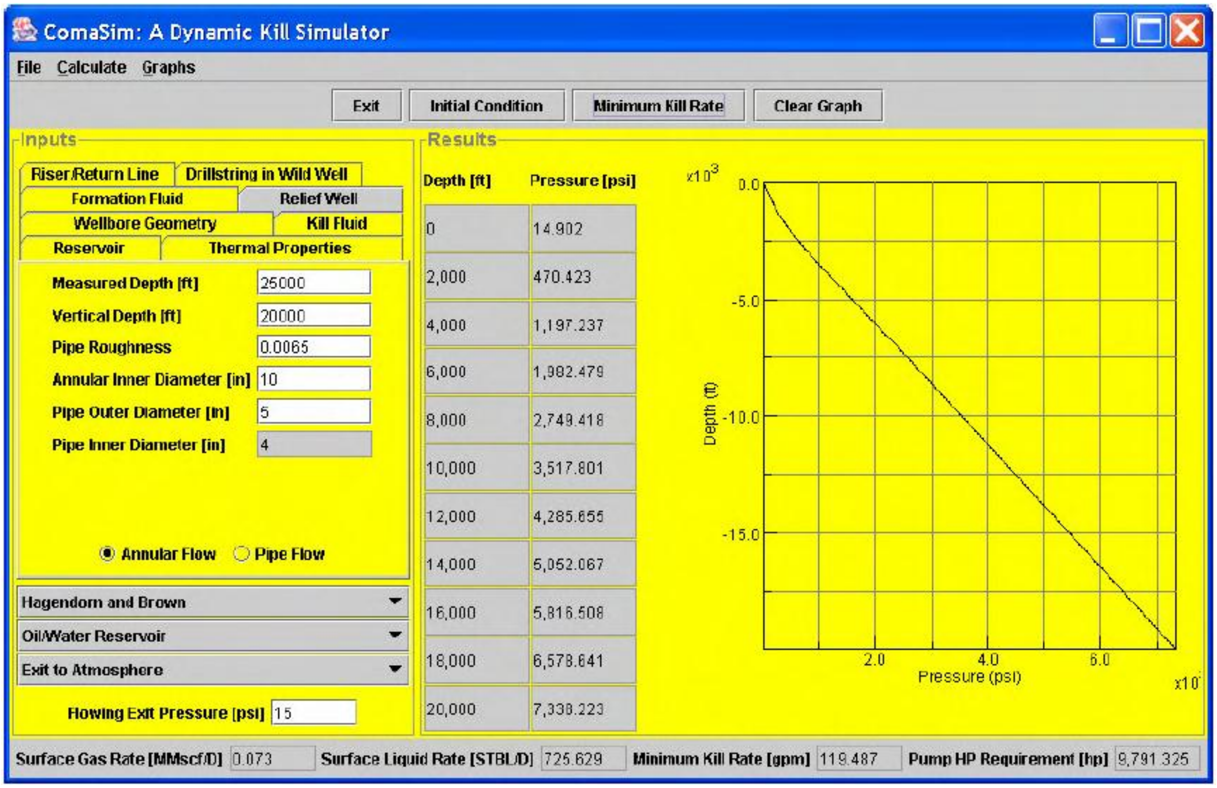


Figure 4-5 - Screenshot of a Single Page in the COMASim Model [43]

The simulator first calculates the initial conditions for the blowout based on multiphase calculations and given inputs. Examples of input values are [43]:

- Drill string: OD, ID, length etc. of the blowing well.
- Wellbore geometry: TVD, casing depth from MSL, casing ID etc. of the blowing well.
- Reservoir properties: Average reservoir pressure, permeability, GOR, water cut.
- Kill fluid: Mud weight, yield point etc.

In addition the user can choose the reservoir type (liquid or gas reservoir) and exit point for the blowout e.g. blowout to surface.

The initial conditions results in an IPR curve for the blowing wellbore, determined based on a chosen multiphase model (Hagendorn and Brown, Beggs and Brill or Duns and Ros). By using Nodal analysis the kill rate is found through successive iterations until convergence. The minimum kill rate for the well presented in Figure 4-7 will be 80 bbl/min.

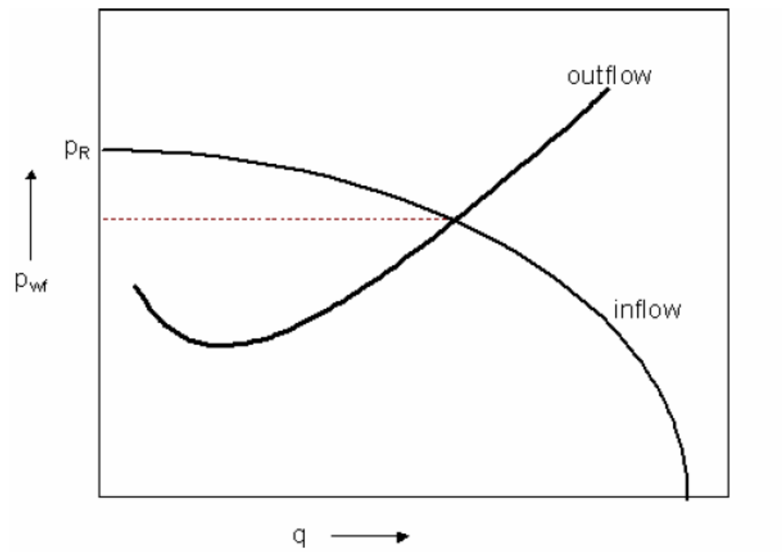


Figure 4-6 - General Nodal Analysis Calculation: Inflow and Outflow Curves [43]

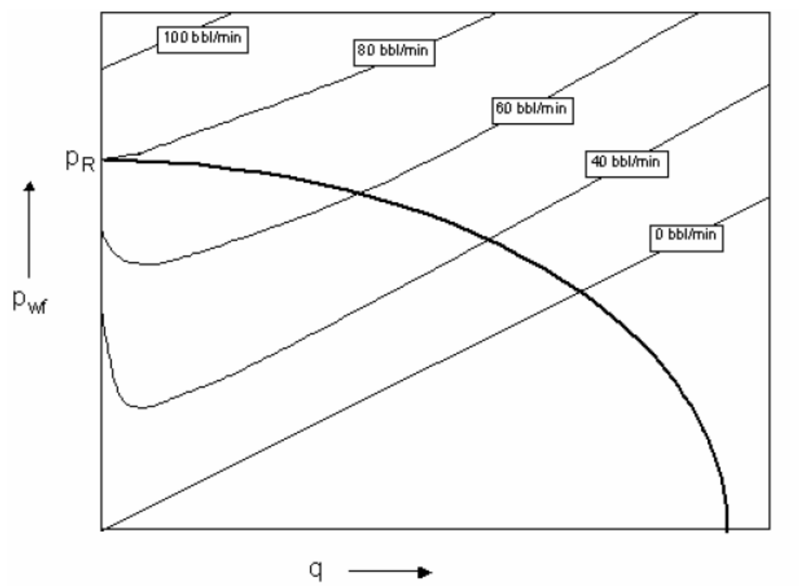


Figure 4-7 - Nodal Analysis [43]

The COMASim simulator does not take into account counter current flow; kill fluid flowing against the blowout flow. The counter current creates a backpressure limiting the blowing flow and reducing the dynamic kill rate needed to kill the well. In other words COMASim has a safety factor; it models to high dynamic kill rates.

COMASim has only been validated theoretically and it is therefore recommended that the model is validated through blowout case histories in the future. [42]

4.6 Other Simulators

On the market today there exist several kick simulators modeling the initial kick phase and the behavior, and risk programs calculating the potentially risk for a blowout to occur. Here a very short description is given.

- WellPlan^{TM} Well Control Software

  Landmark`s Wellplan$^{TM}$ Well Control Software [44] models well kick scenarios, and is a part of the company Halliburton`s software products and services. WellPlan^{TM} uses the well kill methods Driller`s and Wait and Weight to calculate the pressures observed when a kick is taken and circulated out of the wellbore. Afterwards, the software tool performs calculations required for pumping schedules and decision making. The tool also assists in casing design and in planning for contingency during well design phases.

- BlowFAN

  BlowFAN is a risk program supporting the risk assessments of blowouts and is developed (1995) and provided by ScandPower [45, 46]. BlowFAN is used to assess risk in different operations e.g. drilling and intervention operations and planning operations. The output from the BlowFAN software is calculated frequencies of blowouts "*derived specific for well operations and fields of interest* [45]" for different blowout scenarios. BlowFAN uses updated statistics from SINTEF.

  BlowFAN has been used on several fields on the Norwegian continental shelf, some of them are:
  - Kristin (Statoil)
  - Ekofisk (Phillips Petroleum Company)
  - Troll Olje, Njord, Visund, Oseberg (Norsk Hydro)

- Drillbench

  Drillbench [47] is a transient two-phase flow simulator provided by SPT Group [34]. The simulator can perform dynamic simulations for underbalanced drilling, managed pressure drilling, kick scenarios and kick tolerance calculations, well kill operations etc.

  Several applications are included in the simulator and some of them are:

- The Dynamic Well Control application - a tool for decision making and risk evaluation, simulating the flow process in the well in order to obtain well control.
- Kick program - used for well control engineering, training and decision making. The program can evaluate well control procedures and kick detection systems, and train personnel prior to difficult drilling operations.
- Dynaflowdrill - a software application for underbalanced drilling (UBD). The program can define optimal operation parameters or investigate transient scenarios that will occur in an UBD operation (e.g. connections).

5 Development and Use of a Simple Transient Model for Blowout

5.1 Programming of a Transient Two-Phase Model

This section will describe the main steps in the programming of a transient two-phase model for oil and liquid. The basis for the two-phase model is a transient code based on the AUSMV scheme (presented in [20, 31]) simulating two-phase flow of gas and liquid (water). The task is to re-program the transient code for gas and liquid to a transient code suitable for oil and liquid. The main programming steps that will be reviewed in the following are [48]:

- Implementation of a numerical approach in order to find the pressure parameter from the conservative variables.
- Verification of the stability when changing from an analytical to a numerical approach.
- Implementation of a more realistic friction model.
- Implementation of a PI model to account for the reservoirs deliverability and thus the inflow of oil.
- Change of the expression for the velocity of sound to suit a model simulating two liquid phases.
- Implementation of a frictional pressure gradient for the relief well to be able to simulate a dynamic kill operation.

5.1.1. Numerical Approach

First, recall that the conservation equations for a two-phase flow neglecting flow area changes can be written as:

$$\partial t \begin{pmatrix} w_1 \\ w_2 \\ w_3 \end{pmatrix} \partial x \begin{pmatrix} v_l w_1 \\ v_g w_2 \\ v_l^2 w_1 + v_g^2 w_2 + p(w_1, w_2) \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ -q \end{pmatrix} \qquad \text{Eq. (3.4), Section 3.2}$$

The pressure, p, is a variable not found directly from the scheme, but is depending on the updated conservative variables (w_1, w_2). The pressure can be written as $p = p\ (w_1, w_2)$ where $w_1 = \rho_l \alpha_l$ and $w_2 = \rho_g \alpha_g$.

In the original code (gas-liquid), the pressure is found using an analytical approach: The relation between the conservative variables (w_1, w_2) and the primitive variables (pressure, volume fractions and densities) are given by: $w_1 = \rho_l \alpha_l = \rho_l (1 - \alpha_g)$ and $w_2 = \rho_g \alpha_g$. Knowing that $\alpha_l + \alpha_g = 1$ the expression for w_2 can be rewritten to $\alpha_g = w_2/\rho_g$. This leads to the following expression $w_1 = \rho_l \alpha_l = \rho_l(1 - w_2/\rho_g)$ which rearranged gives:

$$w_1 - \rho_l\left(1 - \frac{w_2}{\rho_g}\right) = 0 \tag{5.1}$$

Recall that the equations for the densities of gas and liquid are given by (section 3.3):

- Density of gas (ideal gas law assumed): $\rho_g = \frac{p}{\alpha_g^2}$, where p is the pressure given in $[Pa]$ and the velocity of sound in the gas phase it assumed to be α_g = 316 $[m/s]$.
- Density of liquid: $\rho_L = \rho_{l,o} + \frac{p - p_{l,o}}{\alpha_l^2}$, where $\rho_{l,o}$ $-$ 1000 $[kg/m3]$, $p_{l,o}$- $100\,000$ $[Pa]$, and the velocity of sound in the liquid phase is assumed to be a_l = 1000 $[m/s]$.

Inserting the expressions for the gas and liquid densities into equation (5.1) a quadratic expression for the pressure is obtained, simplified with the use of coefficients (a, b, c).

$$ap^2 + bp + c = 0 \tag{5.2}$$

The pressure, p, can then be determined analytically by calculating the expressions for the coefficients and solving equation (5.2).

When changing the code to oil and liquid the expression for the density of gas in equation (5.1) needs to be replaced with the expression for the density of oil [30]:

$$\rho_o = \rho_{o,0} + \frac{p - p_{o,0}}{\alpha_o^2} \tag{5.3}$$

Here $\rho_{o,O}$ = 800 $[kg/m^3]$, $p_{o,0}$ = $100\,000$ $[Pa]$, and the velocity of sound in the oil phase is assumed to be $a_o = 316$ $[m/s]$.

Equation (5.1) is then rewritten to: $w_1 - \left(\rho_{l,o} + \frac{p - p_{l,o}}{a_l^2}\right)\left(1 - \frac{w_2}{\left(\rho_{o,0} + \frac{p - p_{o,0}}{a_o^2}\right)}\right) = 0$ (5.4)

Expression (5.4) suddenly becomes very difficult to solve analytically; giving more complex coefficients (a, b, c). Hence, a more flexible numerical approach for finding the pressure will be implemented.

In the numerical approach equation (5.1) is rewritten to a function of the pressure:

$$f(p) = w_1 - \rho_l(p)(1 - w_2/\rho_g(p)) \tag{5.5}$$

The correct pressure is the one that gives $f(p) = 0$. Hence, the problem is reduced to finding the "zero" of this function.

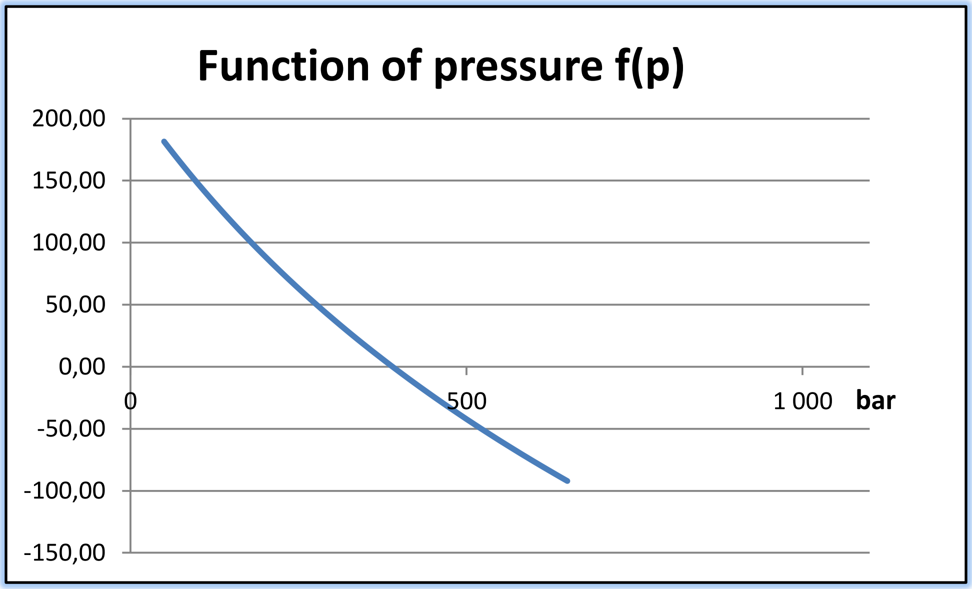


Figure 5-1 - Function of Pressure f(p)[10]

To find the point where $f(p) = 0$ search boundaries must be established to minimize the search time and ensure that a solution can be found. It is important to establish a pressure search interval such that the end points in the interval $[x_2, x_2]$ gives $f(x_1) * f(x_2) < 0$. The values $[f_1, f_2]$ must be a positive and a negative guessed value to ensure that the "zero" of the function is included. The search for the correct pressure is done by halving the search interval always ensuring that $f(x_1) * f(x_2) < 0$ (Bisection method [52]). When $f(p)$ is smaller than the tolerance value, the correct pressure is found. If a solution is not found, the search interval is adjusted and iterated until a solution is reached.

Reference is given to **Appendix A – Two-Phase Code for Oil and Liquid, A1 and A4**.

[10] f(p) is calculated with eq. (5.5). Assumed oil and liquid densities is $800\ kg/m^3$ and $1000\ kg/m^3$, respectively.

5.1.2. Verification of the Stability

After implementing the numerical approach to find the pressure, it was natural to compare it with the previous analytical approach. The gas-liquid system was used. Several simulations were run where both the time step and the flow rates was changed. In the following a short review of the results is given. For further details reference is given to the diagrams in **Appendix B – Results, B1**.

- Varying the size of the time steps:
  The analytical and numerical approaches were stable when changing the size of the time steps, and produced identical results.

  A general observation when changing from the analytical to the numerical approach was that the running time for the numerical approach was longer. This was expected since finding $f\ (p)$ numerically involves more computational steps.

- Changing the inflow rates:
  During simulations with different inflow rates of gas and liquid it was observed that the numerical solution could only handle inflow rates of gas less than $0{,}15\ kg/s$. During this simulation there was approximately 80% gas at the outlet of the pipe with a velocity of $20\ m/s$. Increasing the gas rate towards $0{,}30\ kg/s$ induced a more challenging scenario to model for the numerical solution, which could not handle the increased dynamics. A reason for this may be that the CFL criterion in the last box was close to being broken.

  The conclusion is therefore that the analytical approach is more robust than the numerical approach when it comes to modeling the dynamics in the well with a high gas rate.

5.1.3. Change of Friction Model

The friction model originally used (Section 3.3, eq. 3.10) is very simple and only valid for laminar pipe flow. When converting the code to oil-liquid it was found necessary to both take into account the annular geometry of the well and to distinguish between laminar and turbulent flow. In general, it is important to have good friction models when evaluating blowout kills.

To obtain a more realistic simulation when modeling oil and liquid the following model for the frictional term will be used [31]:

$$F_w = \frac{2f\rho_{mix}v_{mix}abs(v_{mix})}{(d_{out}-d_{in})} \tag{5.6}$$

The expression for the friction factor, f, is determined based on the Reynolds number giving the transition between laminar and turbulent flow [31]:

$$Re = \frac{\rho_{mix} abs(v_{mix})(d_{out} - d_{in})}{\mu_{mix}} \tag{5.7}$$

Where μ_{mix} is the mixture viscosity of oil and water with the following values used in the simulations $\mu_o = 2\ cP$ and $\mu_l = 1\ cP$.

If Re < 2000, the flow is laminar and $f = \frac{24}{Re}$ should be used.

If Re > 3000, the flow is turbulent and $f = 0{,}052Re^{-0{,}19}$ should be used.

For Re in between, interpolation is used.

The friction gradient (F_w) is used in the equation for the source term [20]:

$$q = F_w + F_g = \frac{2f\rho_{mix} v_{mix} abs(v_{mix})}{(d_{out} - d_{in})} + g(\alpha_l \rho_l + \alpha_o \rho_o).$$

The author refers to **Appendix A – Two-Phase Code for Oil and Liquid, A1 and A3** for further details.

5.1.4. Implementation of a Productivity Index Model

The productivity index (PI) can be defined as "*an index expressing the ability of a reservoir to deliver fluids to the wellbore* [4]" and is expressed by the equation:

$$Productivety\ Index\ (PI) = \frac{q}{(P_{res,avg} - P_{well})}$$

To be able to simulate an oil blowout, a PI model is implemented in the transient code to account for the inflow of oil through the wellbore wall; the PI model will deliver a rate of oil in $[kg/s]$ into the well during a certain period of time. The PI value will be gradually increased to avoid a large inflow of oil due to the initial pressure difference between the two fluids. This is done through interpolation.

If a gas blowout was to be simulated instead, eq. 2.14 (section 2.2.2) would be used to describe the inflow of gas.

For further details regarding the PI model reference is given to **Appendix A – Two-Phase Code for Oil and Liquid, A1.**

5.1.5. Velocity of Sound

Based on the findings from the numerical examples performed in [31] the expression for the velocity of sound in the fluid mixture, used in calculating the fluxes in the AUSMV scheme, was changed to:

$$c(\alpha_o) = a_l(1-\alpha_o) + a_o\alpha_o \tag{5.8}$$

Where α_o is the volume fraction of oil and a_o, a_l are the velocities of sound $[m/s]$ in the oil and liquid phase, respectively.

This expression was found to be more robust when performing numerical examples with two liquids (oil and water) [31].

For further details reference is given to **Appendix A – Two-Phase Code for Oil and Liquid, A2.**

5.1.6. Frictional Pressure Gradient for the Relief Well

In a dynamic kill operation frictional pressures will arise in the relief well during pumping of kill fluid. To ensure a sufficient kill rate at the top of the relief well to dynamically kill the blowing well, the frictional forces must be taken into consideration when determining the pump pressure. The pump pressure is dependent of the friction forces in the whole system, both in the blowing well and in the relief well.

Assuming that the well is filled with sea water the following frictional pressure equations for single phase flow is valid [21]:

- Laminar flow: $\left(\frac{dP}{dx}\right)_f = \frac{4}{D}\,\frac{16}{Re}\,\frac{1}{2}\,\rho\, v^2$ (5.9)

  Where: $f = \frac{16}{Re}$ (Fanning friction factor) and $Re = \frac{\rho_w\, v_w\, D}{\mu_w}$, ρ_w - density of water $[kg/m^3]$, v_w - flow velocity $[m/s]$, D - hydraulic diameter of section in relief well $[m]$, μ_w - viscosity of water $[Pa\ s]$.

- Turbulent flow: $\left(\frac{dP}{dx}\right)_f = \frac{4}{D}\, f\, \frac{1}{2}\,\rho\, v^2$ (5.10)

  Where the friction factor is given by $f = C * Re^{-n}$, assuming $C = 0.046$ and $n = 0.2$ (Dukler).

The frictional pressure gradient must be determined for the whole relief well including the kill and choke lines at the top. For the simulation in section 5.2 the relief well is divided into four sections:

- Kill and choke lines.
- Cased part of the relief well.
- Open hole section with drill pipe.
- Open hole section with drill collars.

A frictional pressure gradient is determined for each section where the velocities is calculated based on the chosen inflow rate of water ($q_{l,in}$) in the bottom of the blowing well.

The total frictional pressure gradient for the relief well can then be written as:

$$\left(\frac{dP}{dx}_{f,tot}\right)_r = \sum \frac{dP}{dx}_{f,kill\ \&\ choke\ line} + \frac{dP}{dx}_{f,Cased\ well} + \frac{dP}{dx}_{f,Openhole-DP} + \frac{dP}{dx}_{f,Open\ hole-DC}$$

Together with the pressure gradient for the blowing well, the total pumping pressure (P_{pump}) down the relief well required to dynamically kill the blowing well will be:

$$P_{pump} \geq \left(\frac{dP}{dx}_{f,tot}\right)_r + \left(\frac{dP}{dx}_{f,tot}\right)_b$$

Where: r – relief well, b – blowing well.

For details regarding the frictional pressure gradient for the relief well the author refers to **Appendix A – Two-Phase Code for Oil and Liquid, A1**.

5.2.Simulating a Dynamic Kill

5.2.1. Scenario

The background for the numerical simulation is a model only able to simulate water and oil flow. Hence, the case is somewhat simplified. The way the system is initialized is a bit "tricky". The well is initially assumed to be water filled and horizontal. Then the gravity constant, g, is increased from zero to 9,81 m/s^2 in a period of 100 seconds (the well is raised from horizontal to vertical position). Hence, in all figures showing pressure the sharp increase in the beginning is caused by this. Afterwards, an oil blowout is allowed to evolve before water from the relief well is used to kill the blowout.

The model simulates a vertical 4 000 meter deep well with 9 5/8" casing (ID = 8.53") set at 3600 meters. The casing is followed by an 8.5" open hole section for the next 400 meters. A drill pipe with OD 5" is used, where the last part of the drill string has 160 meters of 6 5/8" drill collars. The pore pressure is 1.27 s.g. (≈ 500 bar) and assumed constant. The blowout will occur at 3 800 m TVD in the area between the open hole section and the drill pipe.

The relief well is drilled to an s-shaped curve with a measured depth of 6 000 m, where 9 5/8" casing (ID = 8.53") is set at 5 500 m MD. The last part of the relief well consists of a 500 m MD open hole section with a diameter of 8.5". A 5" drill pipe with 150 meters of 6 3/4" drill collars is assumed in the well.

The blowing well and the relief well will be modeled as to separate systems. In the relief well it is assumed that the pump rate at the top of the well is the same as the pump rate in the bottom of the well. The flow passage between the relief well and the blowing well is perfect (no friction forces) at the connection point, and the pump rate at the bottom of the blowing well is equal to the pumping rate in the relief well.

During the dynamic kill operation water will be pumped via the kill and choke lines at the top of the relief well and into the annulus between the casing (or wellbore wall) and drill pipe. Both the kill and choke lines have a diameter of 3" and a length of 100 m with a flow rate of $q_{lines} = \frac{q_{l,in}}{2}$, giving a total flow inflow rate of $q_{l,in}$ into the blowing well.

The AUSMV scheme is used to calculate the flow (including the friction pressure loss) in the blowing well. The inlet kill rate is assumed to be the same as the pump rate at the top of the relief well. The frictional pressure loss in the relief well is found using the equations in section 5.1.6. Combining the two frictional losses (relief well and blowing well), an estimate of the pump pressure required is achieved.

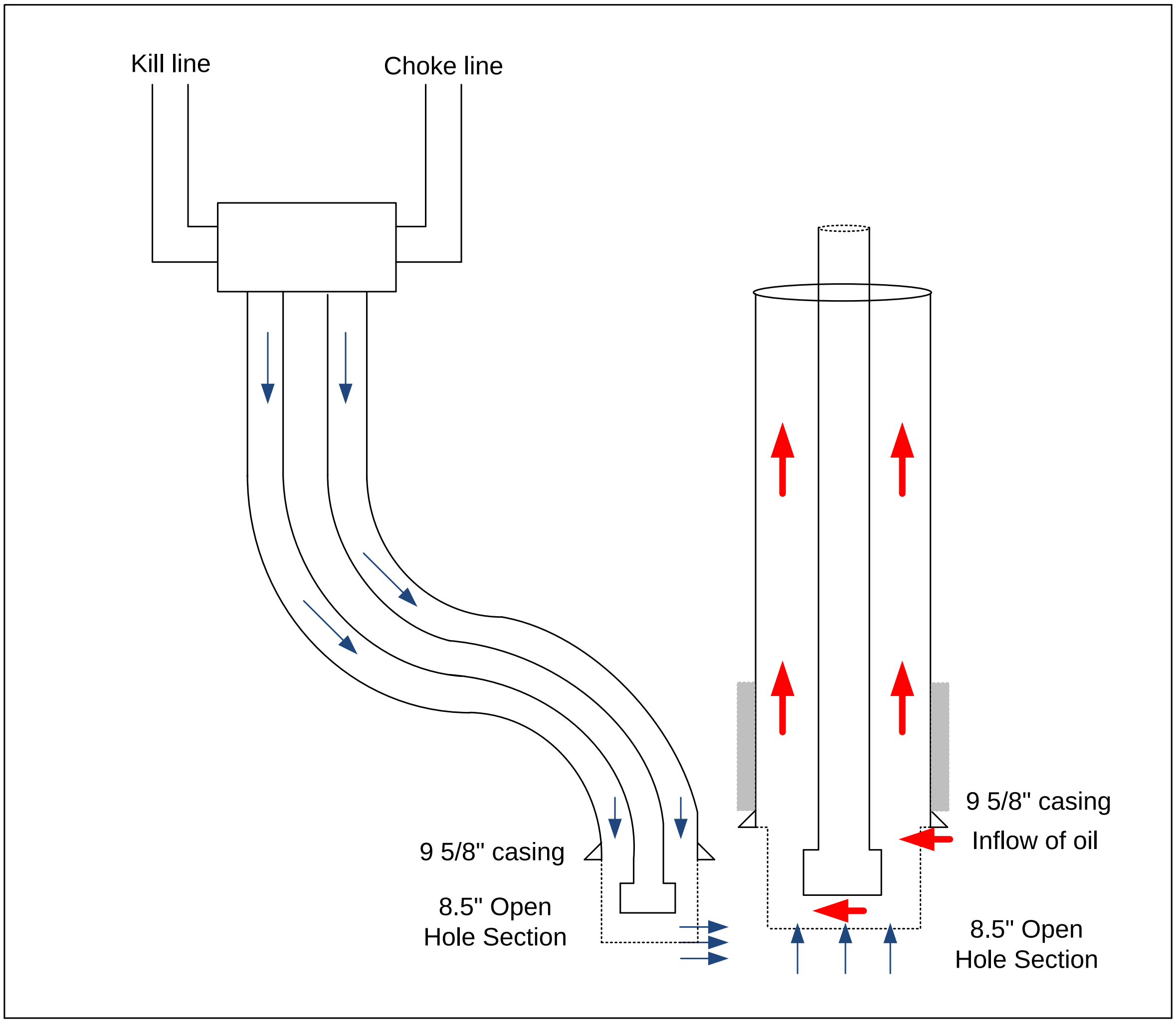


Figure 5-2 - Relief Well and Blowout Well System

5.2.2. Simulation Process

The 4 000 meters deep blowing well is discretized into 50 boxes each with a length of 80 meters. The blowing well will contain only water the first 169 seconds of the simulation. After 170 seconds the PI model will kick-in injecting oil into the well from box number three with an increasing rate. The PI model will operate alone in the well displacing the water and filling the well with oil from 170 - 799 seconds. After 800 seconds the dynamic kill operation will start and water will be injected in the bottom of the blowing well for 500 seconds.

The well will be dynamical killed when:

$$P_{Box\,3} > P_{Res} = 500\ bar$$

5.2.3. Results and Findings from First Simulation Set

In the first set of dynamic kill simulations the velocities of sound for oil and liquid given in [31] was used: $a_o = 316 \left[\frac{m}{s}\right], a_l = 1000 \left[\frac{m}{s}\right]$. One should be aware that these quantities reflect the compressibility of the fluids.

In addition, the viscosities were assumed to be $\mu_o = 2\ cP$ and $\mu_l = 1\ cP$, the PI value was set to $0{,}000001\ kg/s$, the PI model was implemented in box three, and the time step was set to $0{,}02$ seconds.

The simulation was performed with different inflow rates of liquid (water) to find the optimal dynamic kill rate for the blowing well. The results are shown in Table 5-1 below.

Results	$q_{l,in}$ = 100 kg/s (≈ 6 000 liters/min)	$q_{l,in}$ = 150 kg/s (≈ 9 000 liters/min)	$q_{l,in}$ = 160 kg/s (≈ 9 600 liters/min)
Velocities in the relief well:			
$v_{kill,choke\ line}$ [m/s]	10,64	15,96	17,02
$v_{csg,DP}$ [m/s]	4,13	6,20	6,61
$v_{OpenHole,DP}$ [m/s]	4,18	6,28	6,69
$v_{OpenHole,DC}$ [m/s]	7,41	11,11	11,85
P_{pump} [bar]	246,42	393,46	434,2
$P_{fric,b}$ [bar]	131,25	154,26	165,76
$P_{fric,r}$ [bar]	115,17	239,21	268,45
$P_{DynKill}$ [bar]	**499,18**	**500,11**	**509,43**

Table 5-1 - Results from First Simulation

In Table 5-1 $P_{DynKill}$ is the pressure in box number three at reservoir level after a simulation period of 1300 seconds. From the table the minimum optimal kill rate required to dynamic kill the well is $160\ kg/s$.

Additional figures from the simulations are enclosed in **Appendix B – Results, B2**.

Observation 1: Implementation of a PI model

Originally, the PI model was implemented in box number two in the bottom the well. The inflow area in this box was quite small due to the geometry of the 6 5/8" drill collars and the 9 5/8" casing (ID = 8.53") and during the simulation it was observed that the sudden inflow of oil over the small flow area created oscillations in the BHP curve (Fig. 5-3). It was therefore decided to implement the PI model in box number three instead to have an increased inflow area (9 5/8" casing (ID = 8.53") and 5" drill pipe) to inject the oil into, hence reducing the oscillations in the BHP curve (Fig. 5-4).

Figures 5-3 and 5-4 shows the BHP [bar] vs. Time [s] when the PI model is implemented in box number two and three, respectively.

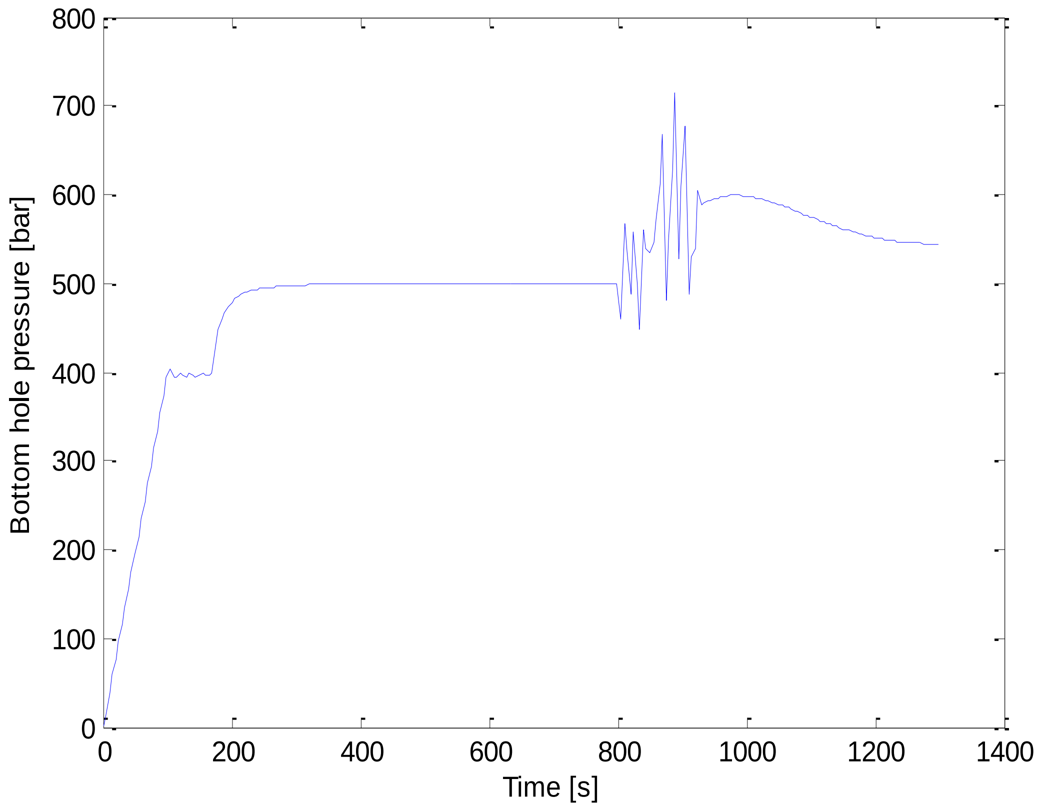


Figure 5-3 - BHP [bar] vs. Time [s], Dynamic Kill Rate: 160 kg/s, for PI Model in Box 2

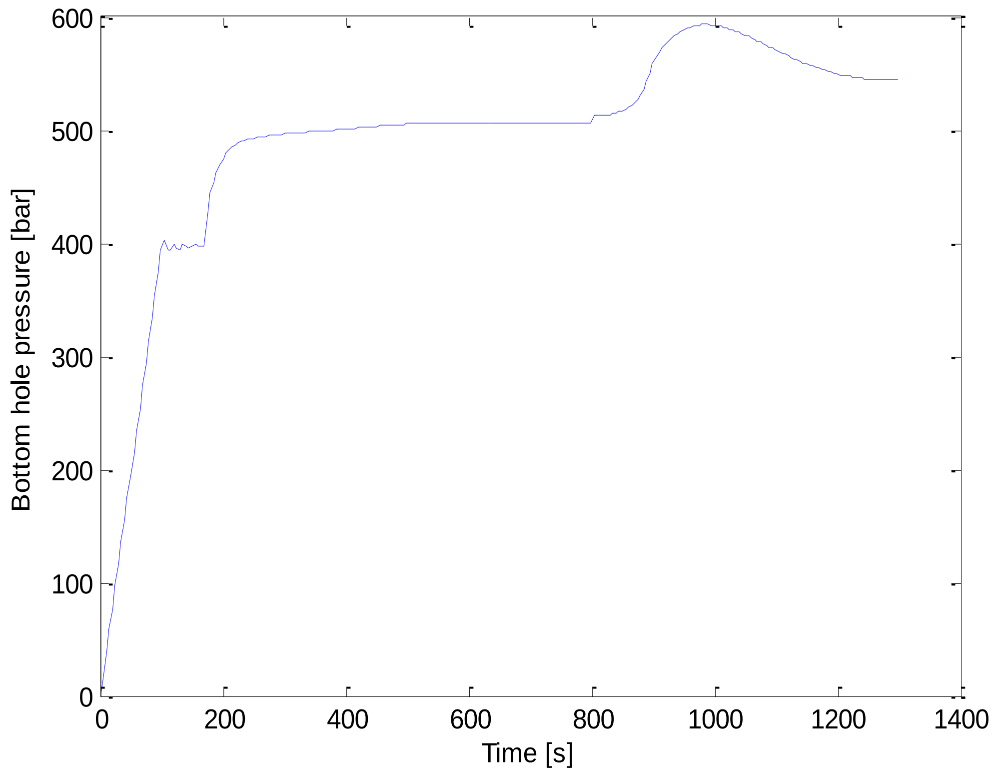


Figure 5-4- BHP [bar] vs. Time [s], Dynamic Kill Rate: 160 kg/s, for PI Model in Box 3

<u>Observation 2: Oil rate out</u>

To ensure that the PI model simulates a realistic blowout it is important to check the oil rate out of the well before the dynamic kill operation starts.

Figure 5-5 gave an oil rate out of the well of:

$$\left(q_{oil,out}\right)_{max} = 34.01\frac{kg}{s} \approx 2\,550.75\ \frac{l}{min} \approx 23\,105{,}5\frac{bbl}{day}$$

Compared to the Macondo accident in the Gulf of Mexico, where approximately 5000 – 40 000 barrels of oil leaked out into the sea each day, it seems that the simulated case represented a "realistic" blowout case with respect to the released rates.

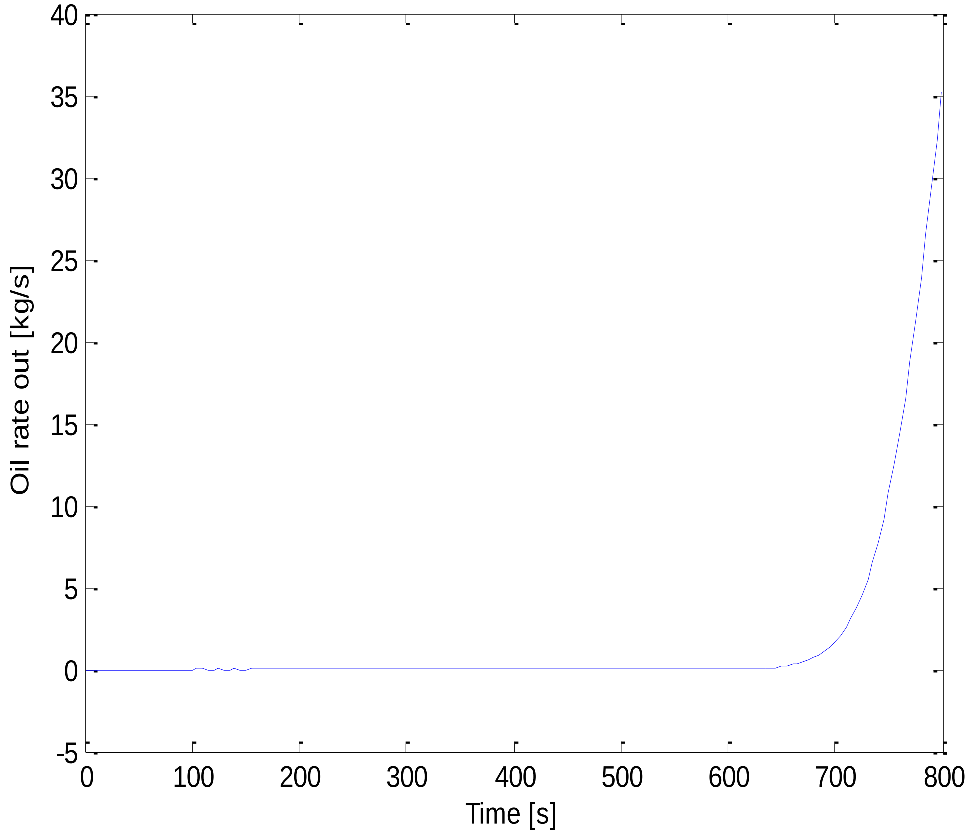


Figure 5-5 - Oil Rate [kg/s] Out of Blowing Well, Dynamic Kill Rate: 100 kg/s

Observation 3: Friction loss gradient

The drill pipe and the drill collars create a reduction in flow area for the kill fluid in the bottom of the well. The frictional forces increase when the flow area is reduced (box 1 - 3), and decrease when the flow area becomes larger (box > 3). This is illustrated in Figure 5-6. The frictional forces also increases with increasing dynamic kill rate, as seen in Figure 5-7. This is normal behavior for the friction loss gradient.

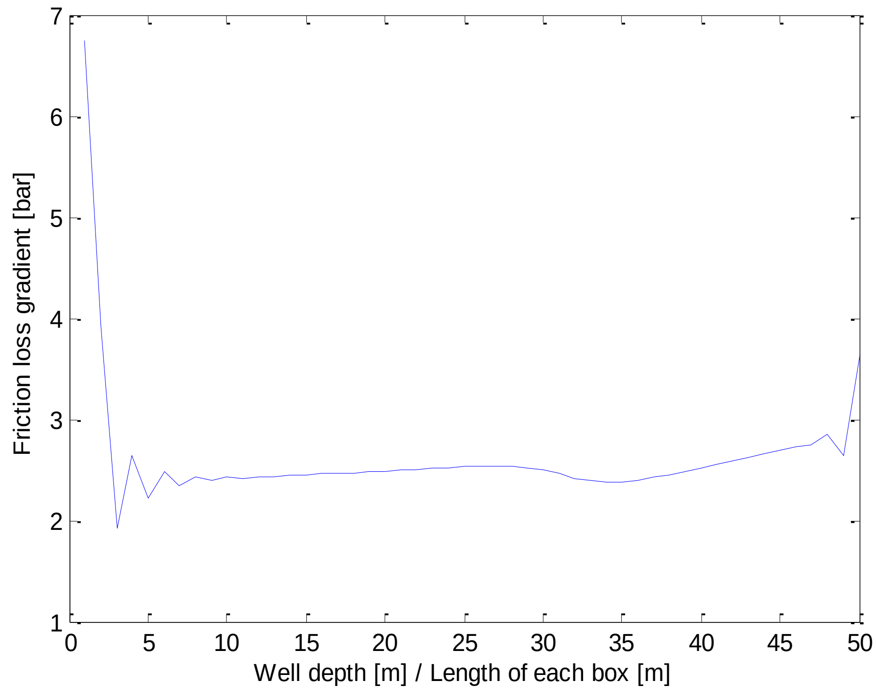


Figure 5-6 - Friction Loss Gradient [bar], Dynamic Kill Rate: 100 kg/s

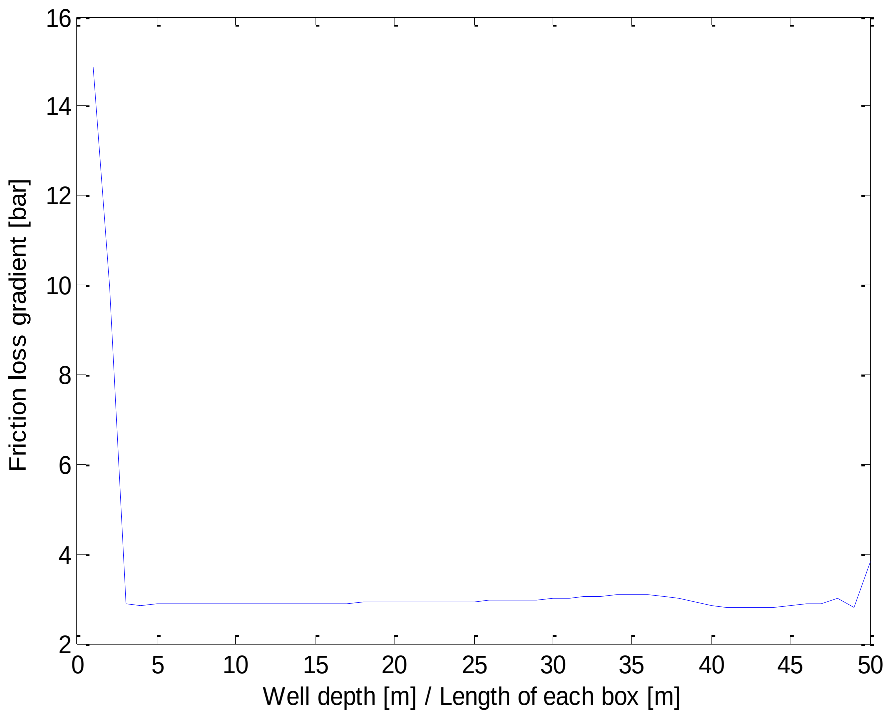


Figure 5-7 - Friction Loss Gradient [bar], Dynamic Kill Rate: 160 kg/s

Observation 4: Oscillations in the PI curve

As explained in section 5.1.4, the PI value is gradually increased. Hence, the PI curve (inflow of oil [kg/s]) gradually increases even though the BHP is also increasing as previously seen in Figure 5-4.

When the dynamic kill starts and kill fluid is injected into the blowing well several oscillations is created in the curve for the PI model, see Figure 5-8. (No oscillation where observed in the BHP after the implementation of the PI model in box number three.) The oscillations are caused by a physical effect; the “struggle” between the PI model and the friction forces created in the well. When the pressure is just below formation pressure, oil will flow and increase well pressure due to friction. The pressure will then increase to above formation pressure, inflow will stop, friction will disappear and the pressure will drop again. This means that there probably is an oscillation in the pressure which causes the PI model to switch on/off. The kill fluid injection rate is not sufficiently large enough to create adequate friction forces in the well to overcome the oil inflow from the PI model straight away. To reduce this physical effect the kill fluid injection rate was increased, see Figure 5-9.

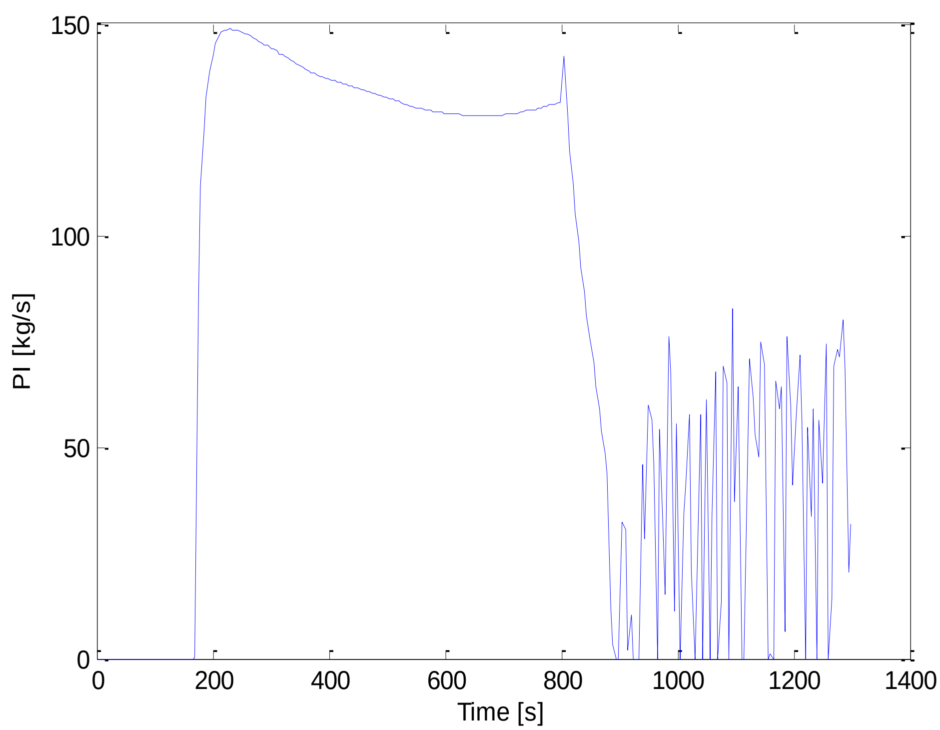


Figure 5-8 - PI [kg/s], Dynamic Kill Rate: 100 kg/s, for PI Model in Box 3

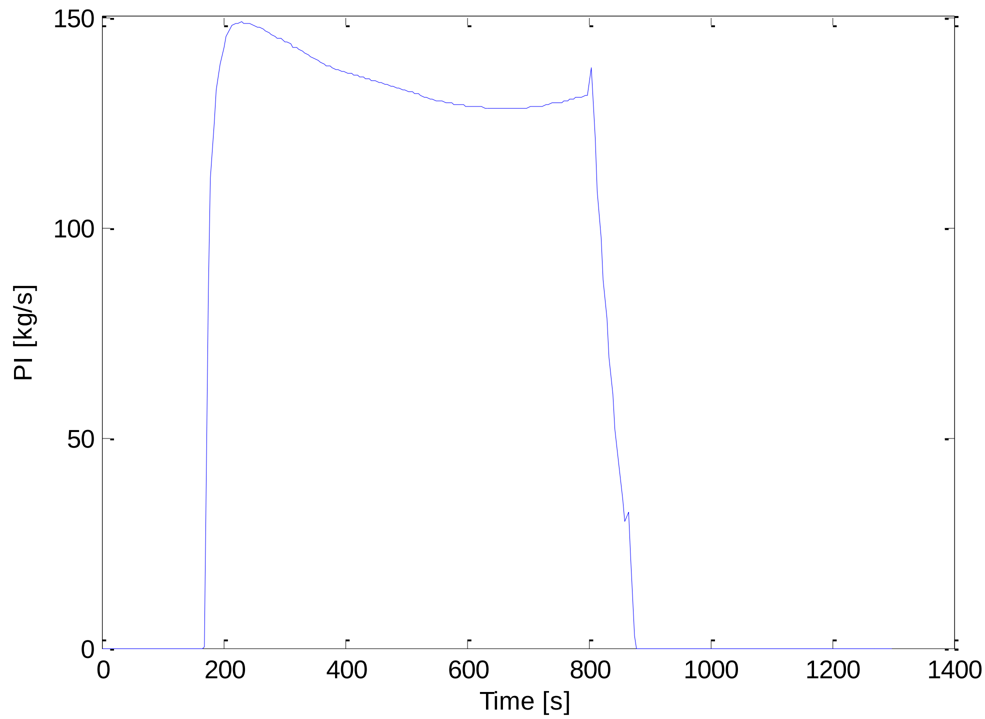


Figure 5-9 - PI [kg/s], Dynamic Kill Rate: 160 kg/s, for PI Model in Box 3

Observation 5: Increasing BHP

When the PI model kicks in after 170 seconds the BHP curve experiences a steep increase due to the displacement of water with oil; a more viscous fluid. After this increase it would be natural to expect that the BHP would decrease due to the reduction in hydrostatic pressure when the oil ($800\ kg/m^3 \approx\ 0.8\ s.g$) is replacing mud, or in this case water ($1000\ kg/m^3\ \approx 1.0\ s.g$). Ideally this means that the pressure in the well when the PI model kicks-in and inflow of oil starts should decrease with:

$0.2\ s.g * 4000\ m * 0.0981 = 78.48\ bar\ \ \approx 79\ bar$ (Friction forces are neglected)

In the transient code for oil and liquid this is not the case. Here the BHP is increasing when the PI model kicks in from 170 – 799 seconds, see Figure 5-10.

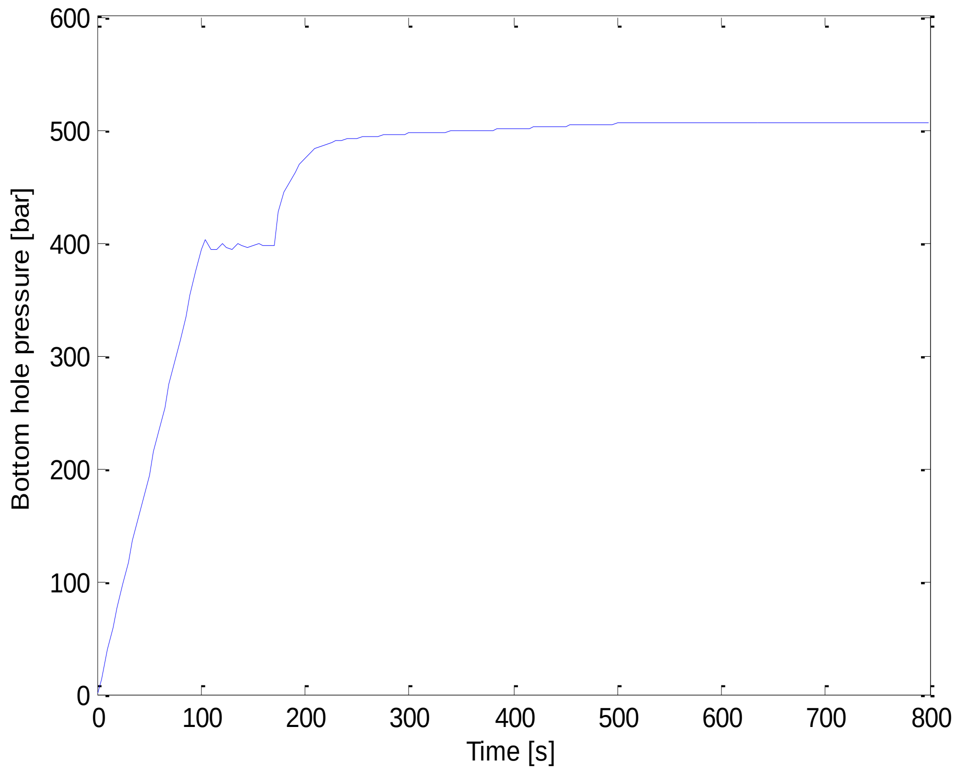


Figure 5-10 - BHP [bar] vs. Time [s], Dynamic Kill Rate: 160 kg/s

The reason for the increase in BHP can be explained by too low velocities of sound for the two phases oil and water (i.e. too high compressibility). It is especially the oil that has very high and unrealistic densities in the lower parts of the well. This causes actually an increase in hydrostatic pressure when oil fills the well. Hence, it was found necessary to change the velocities of sound to more realistic values.

Diagrams showing the oil and water densities vs. depth confirm this conclusion. From Figure 5-11 and 5-12 the densities of oil and water in the bottom of the well are approximately $1300\ kg/m^3$ and $1050\ kg/m^3$, which is higher than the densities used in the simulation ($800\ kg/m^3$ and $1000\ kg/m^3$). The high densities together with frictional forces increase the BHP in the well.

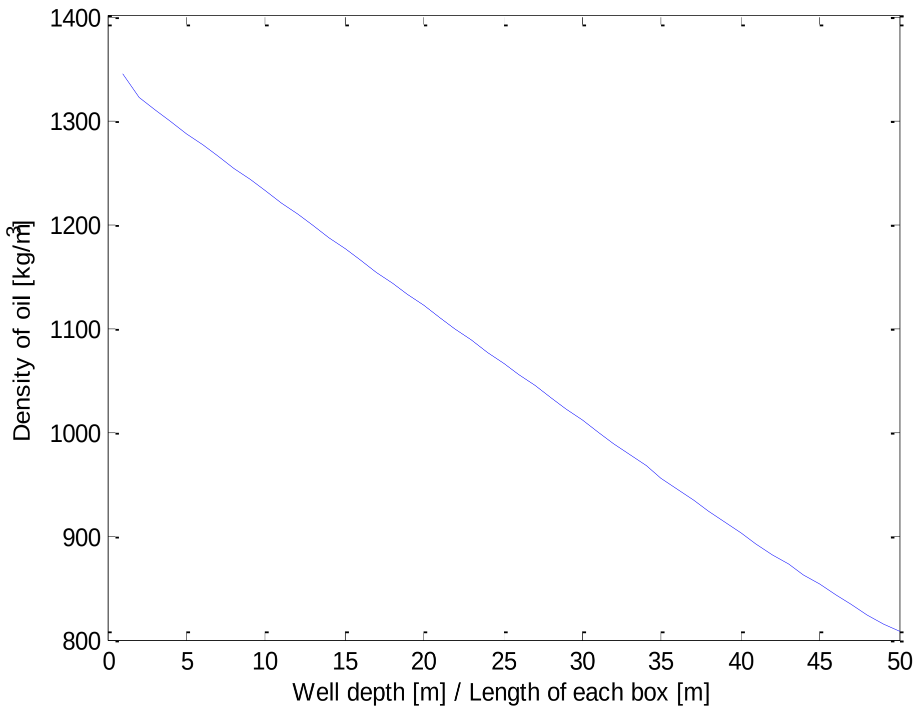


Figure 5-11 - Density of Oil [kg/m^3], Dynamic Kill Rate: 160 kg/s

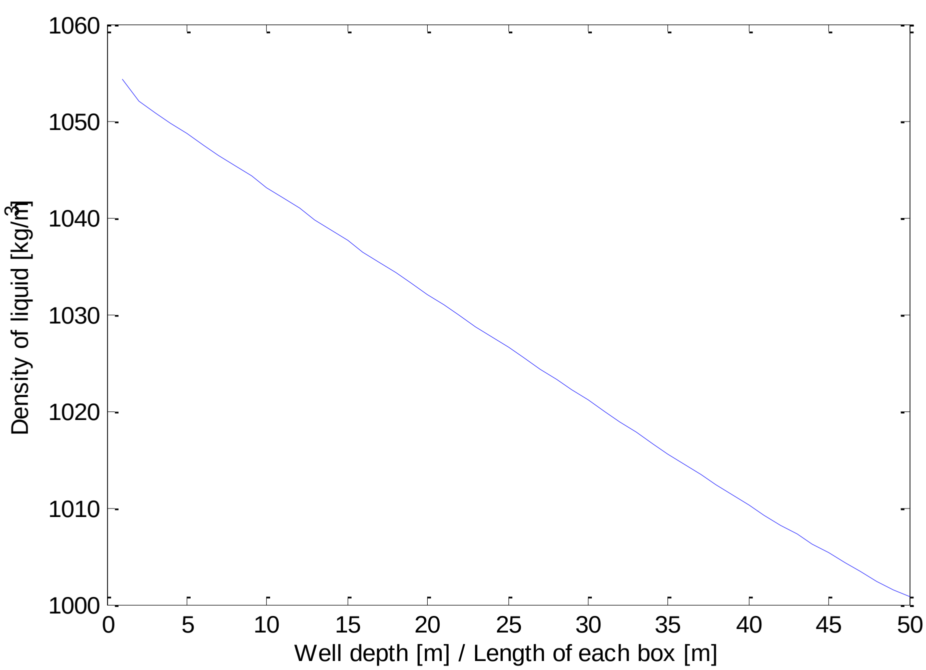


Figure 5-12 - Density of Liquid [kg/m^3], Dynamic Kill Rate: 160 kg/s

5.2.4. Results and Findings from Second Simulation Set

After the first simulation set the velocities of sound for oil and water was increased to $1100\ m/s$ and $1500\ m/s$, to simulate more realistic densities in the bottom of the well. The viscosities ($\mu_o = 2\ cP, \mu_l = 1\ cP$), the PI value ($0{,}000001\ kg/s$) and the location of the PI model (box three) remained unchanged but the time step was reduced to 0,005 seconds for the results given in Table 5-2.

Results from the second simulation:

Results	$q_{l,in}$ = 90 kg/s (≈ 5 400 liters/min)	$q_{l,in}$ = 150 kg/s (≈ 9 000 liters/min)	$q_{l,in}$ = 190 kg/s (≈ 11 400 liters/min)
Velocities in the relief well:			
$v_{kill,choke\ line}$ [m/s]	9,57	15,96	20,21
$v_{csg,DP}$ [m/s]	3,72	6,20	7,85
$v_{OpenHole,DP}$ [m/s]	3,77	6,28	7,95
$v_{OpenHole,DC}$ [m/s]	6,67	11,11	14,07
P_{pump} [bar]	248,90	405,53	602,23
$P_{fric,b}$ [bar]	153,52	166,32	236,41
$P_{fric,r}$ [bar]	95,38	239,21	365,82
$P_{DynKill}$ [bar]	**498,81**	**500,92**	**568,00**

Table 5-2 - Results from Second Simulation

$P_{DynKill}$ (Table 5-2) is the pressure in box number three at reservoir level after a simulation period of 1300 seconds. In the second simulation set the dynamic kill rate was estimated to be $> 150\ kg/s$.

Additional figures from the simulations are enclosed in **Appendix B – Results, B3**.

Observation 1: Realistic densities of oil and water

After increasing the velocities of sound for oil and liquid (water) the densities in the bottom of the well became more normal, approximately $850\ kg/m^3$ and $1025\ kg/m^3$, see Figures 5-13 and 5.14.

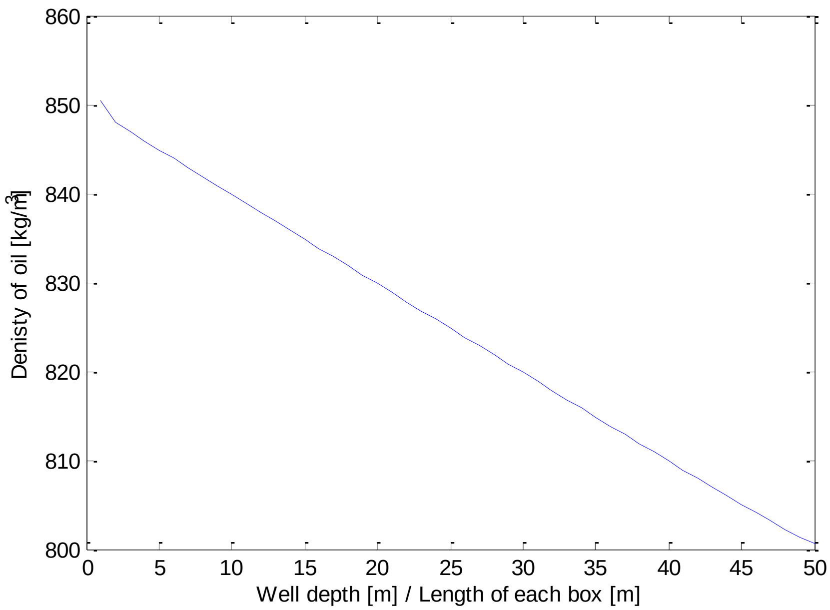


Figure 5-13 - Density of Oil [kg/m^3], Dynamic Kill Rate: 190 kg/s

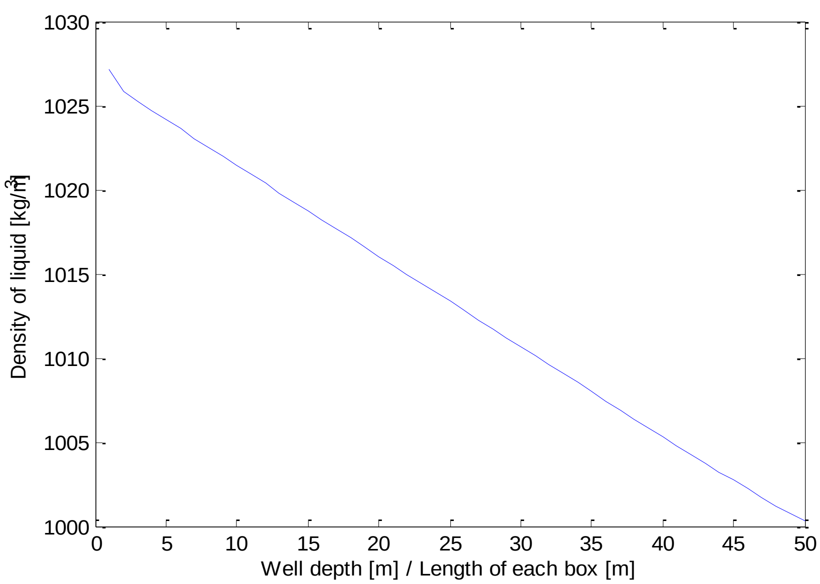


Figure 5-14 - Density of Liquid [kg/m^3], Dynamic Kill Rate: 190 kg/s

Observation 2: Oil rate out

The adjusted densities of oil and liquid in the well affected the PI model and the oil rate out of the well before the dynamic kill operation started.

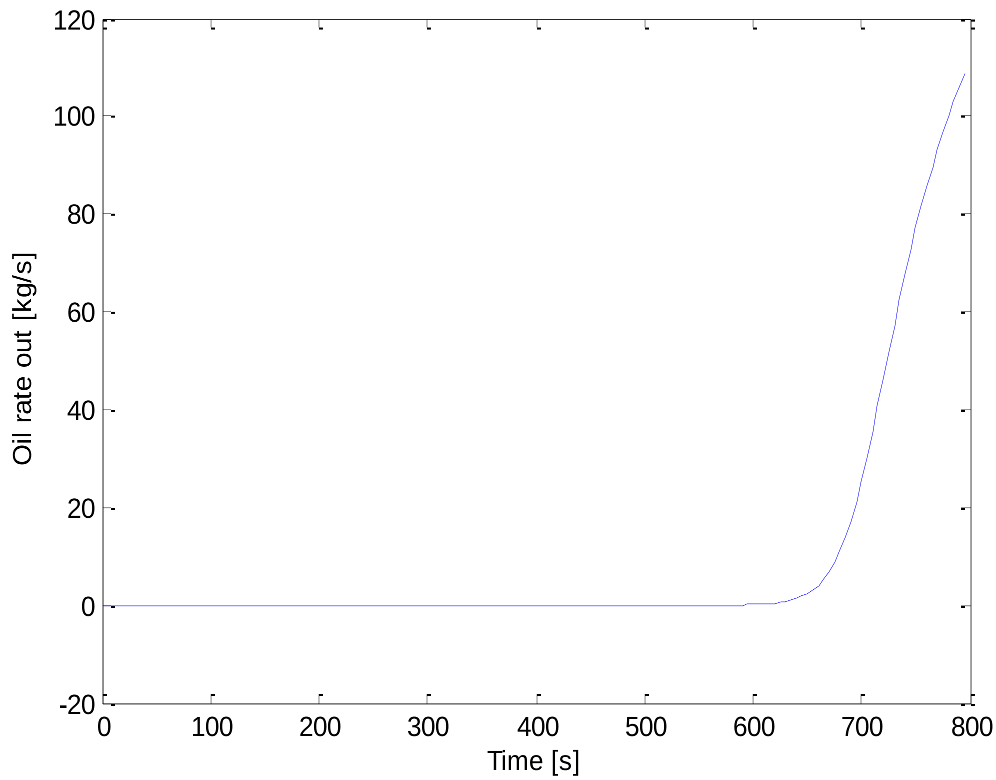


Figure 5-15 - Oil Rate [kg/s] Out of Blowing Well, Dynamic Kill Rate: 190 kg/s

Figure 5-15 gives an oil rate out of the well of:

$\left(q_{oil,out}\right)_{max} = 108{,}7\ \frac{kg}{s} \approx 8\ 152{,}5\ \frac{l}{min} \approx 73\ 848\ \frac{bbl}{day}$, which is large compared to the daily release from the Macondo accident $(5000 - 40\ 000\ bbl/day)$.

The oil rate out of the well has increased due to the changed densities of oil and water, giving a reduced hydrostatic pressure. More friction is therefore needed in order to kill the well.

Observation 3: Oscillations in PI curve

With the increased oil rate oscillations where observed in the PI curve again (but not as serious that it affected the BHP creating oscillations in the curve). First, an attempt was made to increase the PI value more gradually (i.e. trying to make the inflow conditions more smooth) by extending the interpolation interval. In addition, the time step was reduced from 0,02 to 0,005 seconds. This only helped slightly on the oscillations. The oscillations was large when the dynamic kill rate was small (Fig. 5-16), and was reduced when the kill rate was increased (Fig. 5-17). Increasing the kill rate had no effects on the oscillations occurring from 500 – 900 seconds.

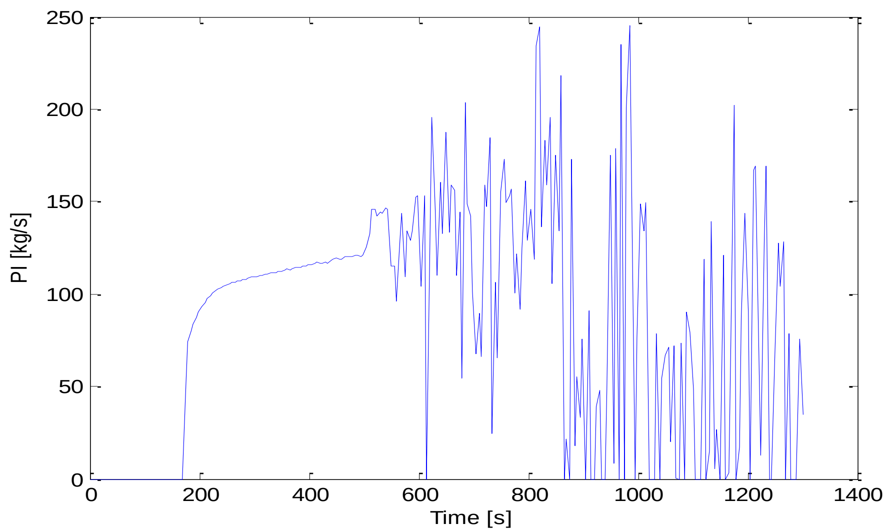


Figure 5-16 - PI [kg/s], Dynamic Kill Rate: 90 kg/s, Time Step: 0.005 sec.

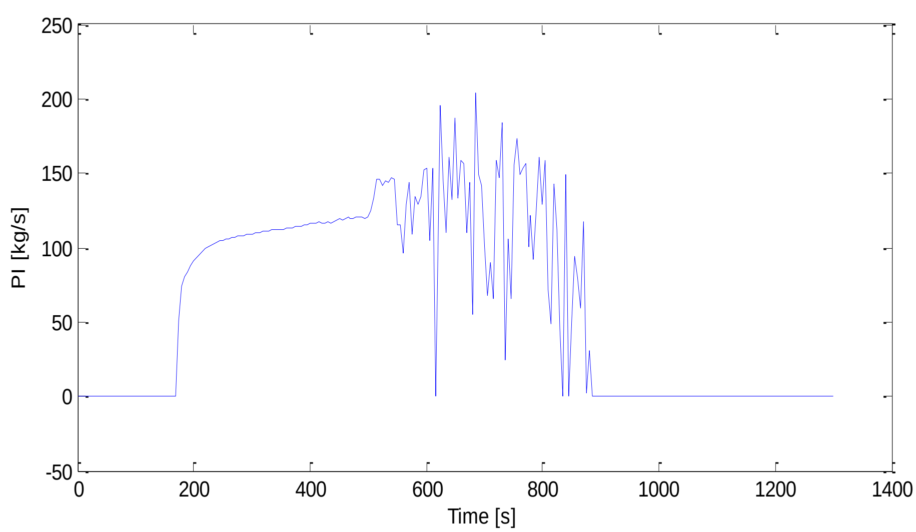


Figure 5-17 - PI [kg/s], Dynamic Kill Rate: 190 kg/s, Time Step: 0.005 sec.

The reason for the oscillations was probably due to the presence of so called stiff source terms [49]. Examples of stiff source terms are e.g. mass transfer of phases reacting on a short timescale. When the well pressure and the reservoir pressure are quite equal and there is a struggle between the PI model and the friction forces, the PI term may act like a stiff source term (switching on/off on a short timescale). When stiff source terms are present a more sophisticated time integration procedure involving small time steps is required. When the time step was reduced far below the CFL criterion, to $0{,}0005$ seconds (Fig. 5.18), the oscillations disappeared. This may confirm that the assumption about the presence of a stiff source term was correct.

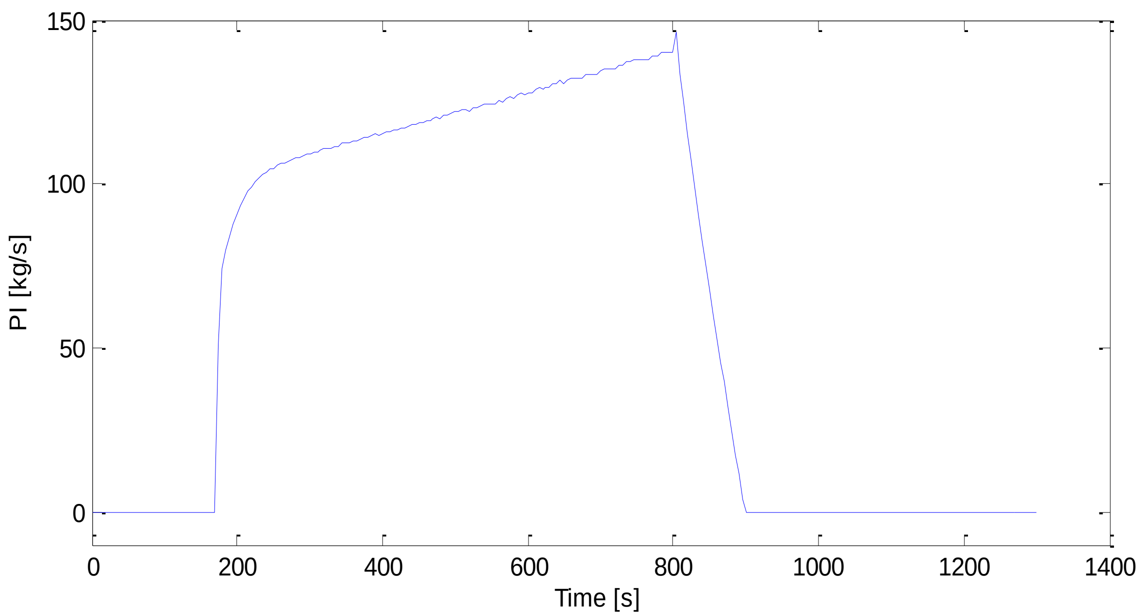


Figure 5-18 - PI [kg/s], Dynamic kill rate: 190 kg/s, Time Step: 0.0005 sec., PI Model in Box 3

Observation 4: Increasing BHP

Despite the adjustments in sound velocities (ref. observation 5, section 5.2.3) the BHP still had a small increase when oil ($\mu_o = 2\ cP$) displaced water ($\mu_l = 1\ cP$) in the well, Figure 5-19. An explanation for the increase is that the well is slightly friction dominated; the increase in friction is larger than the reduction of hydrostatic pressure when oil displaces water. The transient behavior after the well-kill has started is due to changes in friction and density when water is replacing oil. (Water is heavier and also less viscous than oil.)

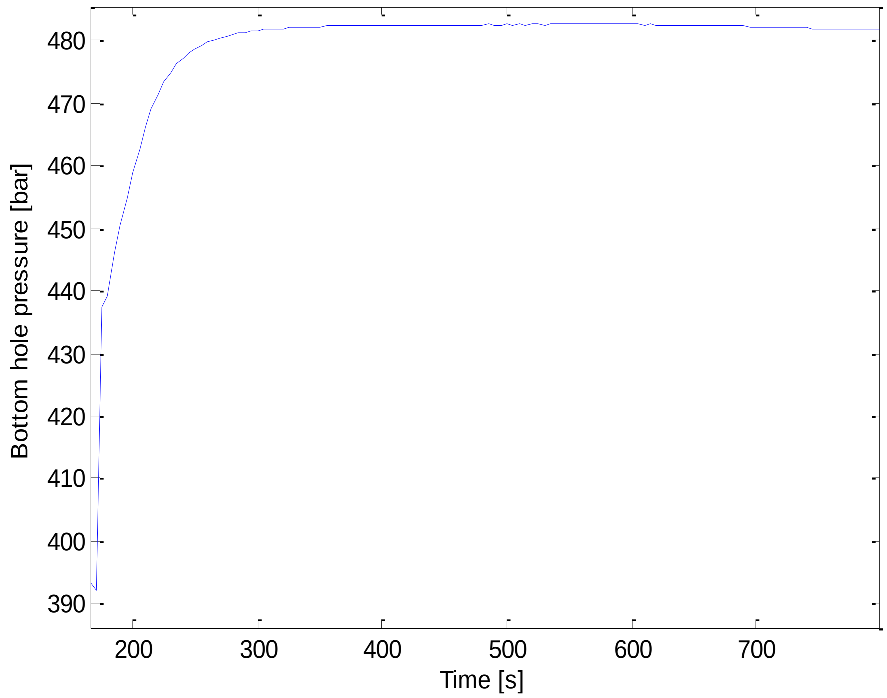


Figure 5-19 - BHP [bar] vs. Time [s], Dynamic Kill Rate: 190 kg/s

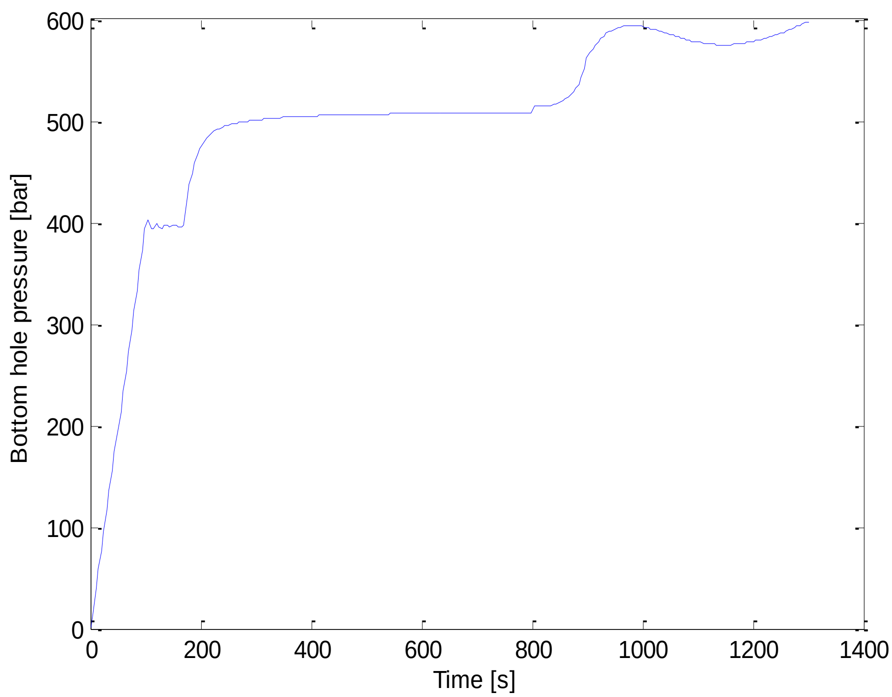


Figure 5-20 - BHP [bar] vs. Time [s], Dynamic Kill Rate: 190 kg/s

Observation 5: Stiff source term verification

To verify that the PI model was acting as a stiff source term; requiring a time step far below the CFL criterion, the PI model was implemented in box number two together with a reduced time step (0,0005 seconds). All other parameters remained unchanged.

Oscillations had previously been observed both in the BHP (observation 1, section 5.2.3) and in the PI curve (observation 4, section 5.2.3) when the PI model was implemented in box two and run with a time step of 0,02 seconds.

With the reduced time step, the oscillations in the BHP - and the PI curve disappeared, ref. Figures 5-21 and 5-22. Hence, this strengthens the conclusion about the PI model acting as a stiff source term. It also gives the opportunity to simulate the inflow of oil though a small flow area (box two), provided that the time step is far below the CFL-criterion.

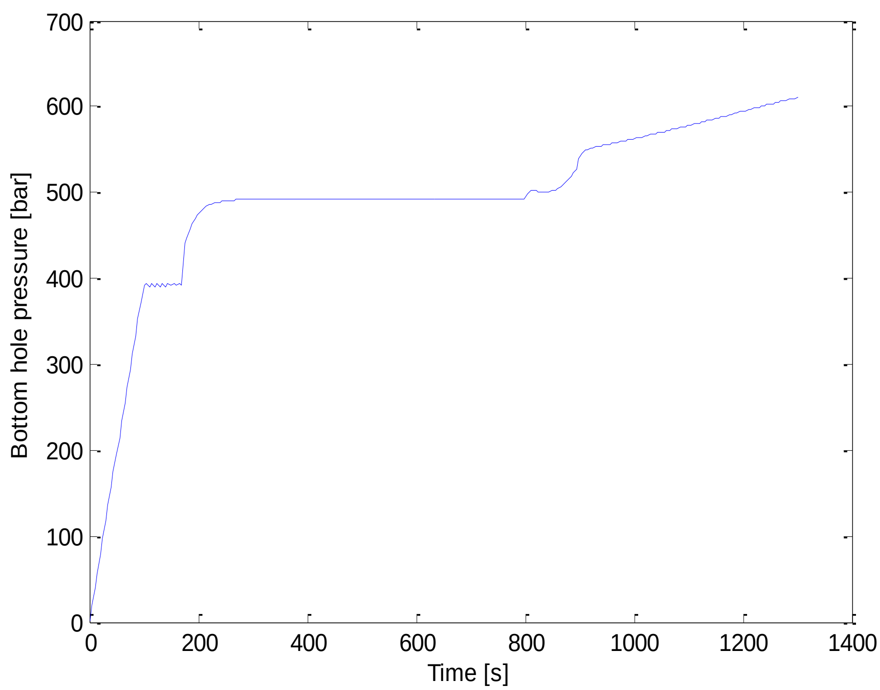


Figure 5-21 - BHP [bar] vs. Time [s], Dynamic Kill Rate: 190 kg/s, Time step: 0.0005 seconds, PI Model in Box 2

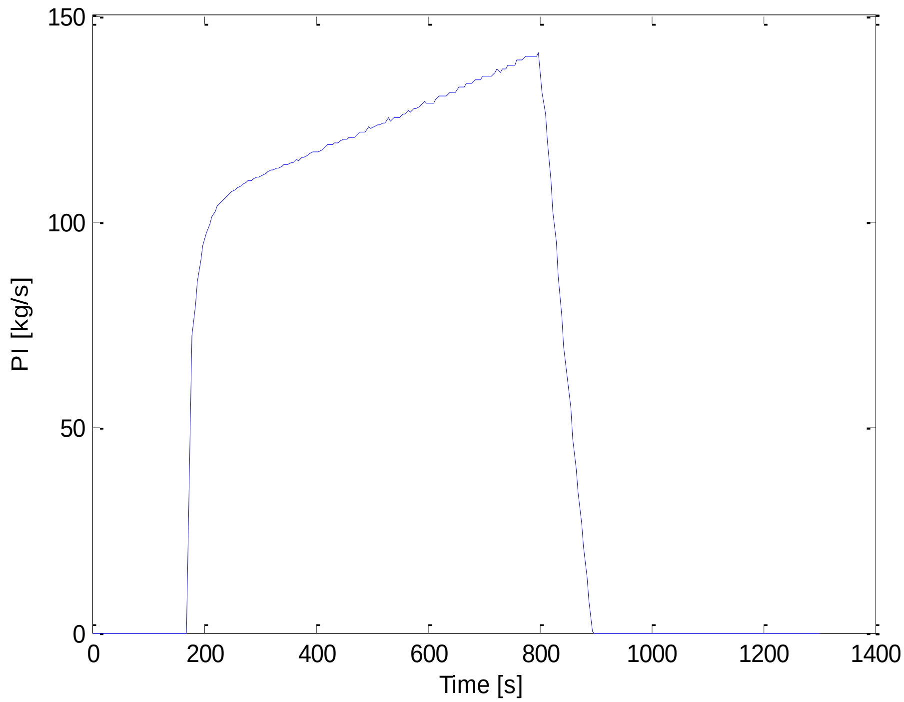


Figure 5-22 - PI [kg/s], Dynamic Kill Rate: 190 kg/s, Time step: 0.0005 seconds, PI Model in Box 2

5.3. Hydraulic Horsepower

The HHP is required to ensure that the pumps deliver enough kill fluid to dynamically kill the blowing well. The total required pumping pressure at the top of the relief well is determined by:

$$P_{pump} = \sum F_r + \sum F_{connection\ channels} + \sum F_b$$

Where: r – relief well, connection channels - the formation between the relief well and the blowing well, and b – blowing well. In the simulation $\sum F_{connection\ channels} = 0$

The hydraulic horsepower can be calculated with formula (2.10) presented in section 2.2.1 based on reference [14], and the results from Tables 5-1 and 5-2. The HHP required for the different kill rates are presented below. Note that the dynamic kill rate ($q_{l,in}$) and the annular frictional pressure $P_{an} = P_{fric,b} + P_{fric,r}$ is converted to bbl/min and psi.

Hydraulic horsepower (HHP)			
First Simulation			
$q_{l,in}$ [bbl/min]	37,74	56,60	60,38
Pan [psi]	3 573,09	5 705,32	6 296,05
HHP	**3 304,30**	**7 912,79**	**9 315,25**
Second Simulation			
$q_{l,in}$ [bbl/min]	33,97	56,60	71,70
Pan [psi]	3 609,05	5 880,19	8 732,34
HHP	**3 004,15**	**8 155,32**	**15 342,03**

Table 5-3 - Hydraulic Horsepower

Leak-off to the formation is not included in these calculations.

5.4.Summary of Modeling and Simulation Work

The following gives a summary of the modeling and simulation work performed in the previous sections.

As mentioned earlier the starting point for the model was a simple transient two-phase model for gas and liquid based on the AUSMV scheme. In order to make the code suitable for the simulation of an oil blowout several programming steps were performed. The first step was to implement a numerical approach to be able to find the pressure parameter in the well from the conservative variables. The gas-liquid code was based on an analytical solution which gave too complex expressions when converting the code to oil-liquid. Hence, implementing the numerical approach gave a more flexible solution. When comparing the analytical and the numerical approaches identical results were obtained, confirming the stability of the implemented numerical method.

To be able to simulate a blowout occurring in a well, the friction gradient term needed to be extended to include both the annular geometry of the well and to distinguish between laminar and turbulent flow. This would provide a more realistic blowout scenario.

The reservoirs deliverability and the inflow of oil into the wellbore were controlled by an implemented PI model. The PI value was gradually increased through interpolation, thus avoiding a large inflow of oil due to pressure differences in the well.

Based on the numerical example performed in [31] the expression for the velocity of sound for the fluid mixture was recommended to be changed when simulating two-phase flow of two liquids. Hence, the expression for the approximate sound velocity was changed when converting the code to oil and liquid.

In order to determine the pump pressure required to dynamically kill the well the friction pressures in the relief well needed to be taken into consideration. This was done through frictional pressure equations for single phase, laminar and turbulent flow. The required pump pressure for the kill operation at the top of the relief well needed to be greater or equal to the sum of the frictional forces in the relief well, connection channels and the blowing well. The pumping pressure was also used to calculate the HHP which is needed to ensure that the pumps can deliver enough kill fluid to obtain a successful dynamic kill.

Several observations were done during the simulations:

- Blowout rate:
  The assumed PI value [kg/s] gave a realistic blowout rate of oil. The simulated dynamic kill represented a realistic blowout case with respect to the released rates.

- Friction loss gradient:
  The friction loss gradient with the implemented friction gradient term (eq. 5.6) operated as expected in the blowing well. The gradient increased when the flow area was small and decreased when the flow area became larger. In addition, the gradient increased with increasing kill fluid rate.

- Densities of oil and water:
  From the first simulation set it was discovered that the model was simulating too high densities for oil and water in the lower parts of the well, causing an increase in the BHP. To simulate a more realistic case the velocities of sound for the two phases were increased from $316\,m/s$ and $1000\,m/s$, to $1100\,m/s$ and $1500\,m/s$. The second simulation set confirmed that more normal densities in the lower parts of the well were being simulated after changing the velocities of sound.

  The increased densities lead to a lower hydrostatic pressure in the well (for the simulations in the second set), hence increasing the oil rate out of the well when the PI model operated alone.

- PI model:
  In the first simulation set oscillations in the PI curve were observed, but after increasing the dynamic kill rate the oscillations disappeared. It was therefore assumed that the pressure in the well was experiencing oscillations affecting the PI model making it switch on/off.

  In the second simulation set the velocities of sound had been increased (making the sound waves travel faster over the boxes), but even with a reduced time step and a high kill rate oscillations were still observed in the PI curve. This led to the assumption that a stiff source term was present. The PI model acted as a stiff source term requiring time steps far below the CFL criterion. To verify the assumption, the PI model was implemented in box two (which had a reduced flow area compared to box three) and the time step was greatly reduced. The results obtained were smooth curves for both the BHP and the PI model, strengthening the conclusion about the PI model acting as a stiff source term.

  Note than the reduced time steps lead to a very large increase in the simulation time.

- Increasing BHP:
  The inflow of oil in the well created an increase in the BHP and the well was assumed to be slightly friction dominated. The friction forces were larger than the reduction in hydrostatic pressure causing an increase in the BHP.

- Hydraulic horsepower:
  In order to determine the required pump pressure (P_{pump}) and the HHP, the frictional forces acting in both the relief well and the blowing well needed to be taken into consideration during the dynamic kill. This was done though the frictional term (eq. 5.6) for the blowing well and through the frictional pressure gradient for the relief well (section 5.1.6). The results were used in estimating the HHP in section 5.3.

6. Uncertainties

6.1.Uncertainties in Two-Phase Models

Generally, two-phase models experience two types of errors [29]: Errors caused by inaccuracy in the closure laws and errors caused by the way the conservation equations are discretized and solved (implicit or explicit). Errors caused by uncertainty in closure laws increases as the complexity of the simulation scenario increases.

The transient two-phase model presented in chapter 5 is based on the AUSMV scheme, a first order scheme. Generally, all first order schemes have challenges with numerical diffusion [18]: The fraction of the flowing phase is averaged between the boxes when moving through the time steps (Δt). The result is that sharp interfaces between phases are smeared out, possibly leading to underestimation of maximum flow rates in transient scenarios. The diffusion can be reduced by using slope-limiter techniques like the MUSCL technique presented in [29]. The AUSMV scheme used as basis in chapter 5 is designed to handle the numerical diffusion and has shown to be a robust first order scheme [31].

The following sections will discuss what is required in a two-phase model to obtain a realistic simulation of a blowout. The sections will have the two-phase model in chapter 5 as reference.

6.2.Realistic Blowout Simulation

Simulating blowouts with two-phase models is a complex task and many parameters must be taken into consideration to obtain realistic results.

For blowouts one can separate between single-phase, two-phase or multi-phase depending on the flowing medium(s). When a blowout develops the flow will first be transient i.e. the flowing medium has an increasing rate, but when fully developed the blowout will be close to a steady-state flow. After some time, depending on the duration of the blowout, the flow will go back to being transient as the well moves towards a killed well situation. To have a fully realistic simulation it is therefore important to have a flow model which can simulate the different flow phases in the blowout and go from transient to steady-state and vice versa.

A blowout can have many different fluid compositions:

- Oil with only small amounts of gas
- Oil with dissolved gas
- Oil, gas and water

In addition there will also be solids or sand particles being transported within the phases. It is important to have a model which can simulate all kinds of fluid compositions. If a blowout consists of oil with dissolved gas it is important to have a good PVT model to predict where the gas boils out in the well. If the PVT model over or under predicts the inflow location of gas, it can lead to calculations with more or less gas in the well than there should be. The amount of gas will greatly affect the hydrostatic pressure in the well, thus affect the dynamic kill.

Several flow regimes will be present in the well when a blowout develops. In the start of the blowout there may be slug or churn flow, but as the velocity of the blowing fluids starts to increase the flow will go towards dispersed bubble or annular flow in the upper part of the well. The transitions between the regimes are not always clearly defined as many different flow regimes can coexist in the wellbore. To get a realistic result of the pressure gradient in the well (hydrostatic pressure, friction and acceleration) both the flow regimes and the transitions should be modeled. For each of the flow regimes there will be a set of closure laws e.g. a slip – and a friction model.

The flow regimes are also affected by the geometry of the wellbore (horizontal, vertical, OD, ID etc.) and the blowout flow configuration: The blowout can arise in the annulus, in the drill pipe or at the outside of the casing, and exit at the surface (drill floor) or subsea, or not exit at all and be an underground blowout. To make the model more versatile several wellbore geometries and flow configurations should be included.

Two-phase models also give results of concern for environmental decisions. The blowout rate for a worst case scenario is important to establish to have an indication of the extent of the blowout. A PI model based on real reservoir data will give a good indication of the blowout rate.

The two-phase model must include all of the above to describe a realistic blowout scenario and should be tested against real case data to confirm its validation. The construction and testing of a two-phase model can be time consuming, but the gain if the accident should occur is huge if contingency plans are pre-developed for worst case scenarios.

6.3. Transient Two-Phase Model for Oil and Liquid

Several assumptions and simplifications have been done in the transient two-phase model:

- Wellbore geometry and configuration:
  The blowing well and the relief well is modeled with a vertical geometry and a blowout configuration in the annulus between the drill pipe and the casing. The wellbore geometry and configuration assumed is realistic, but simplified with regards to the relief wells geometry which in theory is s-shaped. The geometry of the relief well, the leak-off to the formation and the friction forces in the formation between the two wells should be included to make the scenario more realistic.

- Flowing phases:
  The scenario modeled assumes that the blowing well is water filled before the oil blowout develops. A more realistic scenario would be to initially have a heavy mud in the well, but unfortunately the code is not designed to handle this yet.

- Decrease in reservoir pressure:
  After the blowout has reached its maximum point the pressure in the reservoir will eventually start to decrease giving a reduced blowout rate. This decrease in reservoir pressure is not modeled in the code, but will affect the pressures and flow rate in the long run. The simulations in this thesis would not be affected by this due to the short simulation period.

- No-slip condition:
  No-slip condition between the two phases is assumed meaning that the oil and water is flowing with the same velocity. The oil is lighter than the water and there is expected to be some slippage.

- Fluid leak-off:
  Leak-off to the formation between the relief well and the blowing well is not taken into consideration in the simulation, because the wells are modeled as two separate systems. In reality the leak-off is of importance because it affects the pump pressure and the dynamic kill rate required to kill the well.

- Flow regimes:
  Flow regimes are not modeled in the transient code, but laminar and turbulent flow is taken into consideration during calculation of friction loss gradients. If the accuracy of

the closure laws in the model should be improved, models for flow regimes and transitions must be included in the code.

- Counter current flow:
  Counter current flow is not taken into consideration during the dynamic kill operation. This is because the blowout is pure oil with no gas. If a gas blowout is to be modeled counter current should be taken into consideration to avoid modeling slightly too high dynamic kill rates.

- Temperature:
  The temperature development in the wellbore and the surroundings is not modeled in the code. The temperature is important because a considerable amount of heat is transported from the reservoir an up to the wellhead during a blowout. The temperature increase makes the fluids behind the casing expand and the strength of the casing to decrease. In worst case the result could be burst or collapse of casing.

7. Conclusion

Dynamic kill simulators are a valuable tool for the planning- and evaluating phases in a dynamic kill operation, and assist in predicting the feasibility of the dynamic kill attempt. The models give the opportunity to evaluate the dynamics and the operational parameters in a well during a given critical situation. Factors of uncertainty and operational phases that requires special attention can be evaluated in advance as a part of the contingency planning. This will ensure that the personnel are better prepared and aware during the operation. The models can also be beneficial for less experienced personnel as a part of e.g. training programs.

One of the greatest challenges when using simulators is to make the correct assumptions. During a blowout many factors are unknown; flow path, exit point, leak-off from kill fluid etc. For the models to match real-conditions and give realistic results the right assumptions must be made based on real reservoir data; the models are not better than the data and assumptions that are put into them.

In this thesis a transient two-phase model based on the AUSMV scheme is constructed for the simulation of a dynamic kill. The programming steps have been reviewed in detail showing some of the transient effects involved in a dynamic kill and what type of results it can provide. The starting point for the model was a simple transient two-phase code for gas and liquid. In order to make the code suitable for the simulation of a blowout several programing steps where performed:

- Implementation and verification of a numerical approach
- Change of friction model
- Implementation of a PI model to simulate inflow of oil
- Change of the expression for the velocity of sound for the fluid mixture
- Implementation of frictional pressure gradient for the relief well

After running several simulations with different kill rates and observing the results the model proved to be sufficient in describing a dynamic kill operation for a simplified oil blowout. The simulations revealed that the model contained a stiff source term, the PI model, switching on/off on a short timescale creating oscillations in the curve. To avoid the oscillations the model required a more sophisticated time integration procedure involving small time steps far below the CFL criterion.

In addition to the discovery of the stiff source term the blowing well was also found to be slightly friction dominated, i.e. the increase in friction was larger than the reduction of the hydrostatic pressure when oil displaced water.

The transient model is simplified and for future use as a versatile tool some adjustments are recommended. The model should be extended to include several phases; gas, oil, water, and kill fluids (e.g. mud, brine). Here one should also add models to account for the phase transitions and different densities of the kill fluid. This will make the model more adaptable to different blowout scenarios. By modeling the kill fluid with different densities the model will also be able to describe the last steps in a dynamic kill operation which would be to induce heavier mud in the well after the well is killed to fully stabilise it.

Friction is very important for a successful dynamic kill and accurate modeling is essential. Models for flow regimes and transitions should therefore be included. The mechanistic model developed by Lage and Time [28] (presented in section 3.4) is recommended to be implemented in the AUSMV scheme for the oil-liquid code. With the mechanistic model the flow regimes in each box will be determined; hence the correct type of slip and friction model will be calculated based on the flow regime present in the actual box. By doing this the accuracy of the model will be greatly increased.

Future recommendations for the code also include the implementation of models for different blowout configurations, wellbore geometries and flow area changes, providing more realistic results for complex cases. Finally, temperature and decrease in reservoir pressure should be modeled if the simulation period is to be extended.

References

1. Article available from website: Encyclopedia of Earth – Articles – "Deepwater Horizon Oil Spill": http://www.eoearth.org/article/Deepwater_Horizon_oil_spill?topic=50364_ - Visited 01.03.11
2. Report: "Ulykken i Mexicogolfen – Risikogruppens vurdering, 29.11.10", Available from: http://www.regjeringen.no/upload/MD/Vedlegg/hav_vannforvaltning/Forvaltningsplanen_Barentshavet/rapporter/Ulykken_i_Mexicogolfen_Risikogruppens_vurdering_101129.pdf - Downloaded 10.01.11
3. "Deepwater Horizon Accident Investigation Report, 8th of September 2010"- BP, Available from:http://www.bp.com/liveassets/bp_internet/globalbp/globalbp_uk_english/incident_response/STAGING/local_assets/downloads_pdfs/Deepwater_Horizon_Accident_Investigation_Report.pdf - Downloaded 10.01.11
4. Schlumberger Oilfield Glossary: http://www.glossary.oilfield.slb.com
5. Rich. H. Westergaard, "All About Blowout", Senter for Industriforskning Oslo, Norway, - 1987
6. Schubert, Valko, Jourine, Oskarsen, Noyunaert, Meyer, Walls and Weddle, "Development of a Blowout Intervention Method and Dynamic Kill Simulator for Blowouts occurring in Ultra-Deepwater", December 2004. Available from: http://www.boemre.gov/tarprojects/408/Project%20408%20Final.pdf - Downloaded 10.01.11
7. NORSOK Standard D-010, Rev. 3, August 2004
8. SINTEF Offshore Blowout Database, webpage: http://www.sintef.no/Home/Technology-and-Society/Safety-Research/Projects/SINTEF-Offshore-Blowout-Database/ - Visited 10.01.11
9. W.C. Goins, Jr. Riley Sheffield, "Blowout Prevention – Practical Drilling Technology Volume 1", Gulf Publishing Company, Houston, Texas, - 1983, p. 27-31.
10. Samuel F. Noynaert and Jerome J. Schubert, "Modeling Ultra-Deepwater Blowouts and Dynamic Kills and the Resulting Blowout Control Best Practices Recommendations" – SPE/IADC 92626, SPE/IADC Drilling Conference Amsterdam, The Netherlands, 23-25 February 2005.
11. Paola Blotto, Mauro Tambini, Edorado Dellarole, Michele Bonuccelli, "Software Simulation and System Design of Dynamic Killing Technique" – SPE 90427, SPE Annular Technical Conference and Exhibition Houston, Texas, USA, 26-26 September 2004.
12. Alvaro f. Negrao, Victor Gerardo Vallejo-Arrieta, Adam T. Bourgoyne, John Rogers Smith, "Dynamic Kill of Underground Blowouts" – Part of 1994-1999 LSU/MMS Well Control

Research Project. Available from: http://www.boemre.gov/tarprojects/008/008DO.PDF - Downloaded 11.01.11

13. SINTEF Website: http://www.sintef.no/Home/Technology-and-Society/Safety-Research/Projects/SINTEF-Offshore-Blowout-Database/ - Visited 20.01.11
14. Blount, E.M. and Soeiinah, E. "Dynamic Kill Controlling Wild Wells in a New Way", World Oil, October 1981.
15. Picture taken from article in the newspaper "Aftenbladet", available from: http://www.aftenbladet.no/energi/olje/1265046/-__Mange_datafeil_paa_Deepwater_Horizon.html – Downloaded 27.01.11
16. Article taken from AddEnergy website, available from: http://www.addenergy.no/well-control-response-blowout-contingency-planning/olga-well-kill-article389-518.html - Downloaded 08.02.11
17. Petroleum Safety Authority`s website, available from: http://www.ptil.no/about-us/category89.html - Downloaded 27.01.11
18. Kjell Kåre Fjelde, Rolv Rommetveit, Antonio Merlo and Antonio C.V.M Lage, "Improvements in Dynamic Modeling of Underbalanced Drilling" – SPE 81636, Presented at the IADC/SPE Underbalanced Technology Conference and Exibition Houston, Texas, 25-26 March 2003.
19. Thomas J. Danielson, Lloyd D. Brown and Kris M. Bansal,"Flow Management: Steady-State and Transient Multiphase Pipeline Simulation" - OTC 11965, Presented at the 2000 Offshore Technology Conference, Houston, Texas, 1-4 May 2000.
20. Kjell Kåre Fjelde, Steinar Evje, "Hybrid Flux-Splitting Schemes for a Two-Phase Flow Model", Journal of Computational Physics **175**, 2002, p. 674-701.
21. Rune W. Time, "Two-Phase Flow in Pipelines – Course Compendium with Matlab examples and problems", Faculty of Science and Technology, University of Stavanger, January 2009, p. 11-38, 67-99, 106-114, 137.
22. Antonio Carlos V.M Lage, "Two-phase Flow Models and Experiments for Low-Head and Underbalanced Drilling", Dr.Ing. Thesis, Stavanger University College, - 2000.
23. Juan Carlos Mantecon, SPT Group, "The Virtual Well: Guidelines for the Application of Dynamic Simulation to Optimize Well Operations, Life Cycle Design, and Production", SPE Annular Technical Conference and Exhibition, Anaheim, California, USA, 11-14 November 2007.
24. Kjell H. Bendiksen, Dag Malnes, Randi Moe and Svend Nuland, "The Dynamic Two-Fluid Model OLGA: Theory and Application", SPE 19451 – May 1991.
25. AddEnergy homepage: www.addenergy.no
26. Brochure from SPT Group, available from: http://www.sptgroup.com/en/Products/olga/Multiphase-Flow-Simulator/ - Downloaded 08.02.11

27. David Watson, Terry Brittenham and Preston L. Moore, "Advanced Well Control" – SPE Textbook Series Vol. 10, Texas U.S.A, 2003, p. 338-344.
28. Antonio C.M.V Lage, Rune W. Time "Mechanistic Model for Upward Two-Phase Flow in Annuli" - SPE 63127, Presented at the 2000 SPE Annular Technical Conference and Exhibition, Dallas, Texas, 1-4 October 2000.
29. R.J Lorentzen, K.K Fjelde, "Use of Slopelimiter Techniques in Traditional Numerical Methods for Multi-phase Flow in Pipelines and Wells", International Journal for Numerical Methods in Fluids, 2005, **48**: p.723-745.
30. G.E Kouba, G.R MacDougall and B.W. Schumacher, "Advancements in Dynamic Kill Calculations for Blowout Wells" – SPE Drilling and Completion, September 1993.
31. Kjell Kåre Fjelde and Steinar Evje, Work note: "The AUSMV Scheme – A Simple but Robust Model for Analyzing Two-Phase Flow", University of Stavanger, 17 August 2010.
32. T.Flaaten, "Hybrid Flux Splitting Schemes for Numerical Resolution of Two-Phase Flows", Dr.Ing Thesis, NTNU, Norway 2003.
33. L.W Abel and D.W Shackelford, "Comparison of Steady State and Transient Analysis Dynamic Kill Models for Prediction of Pumping Requirements" – IADC/SPE 35120, Presented at the 1996 IADC/SPE Drilling Conference in New Orleans, Louisiana, USA, 12-15 March 1996.
34. SPT Groups homepage, available from: www.sptgroup.com
35. SINTEF`s homepage, available from: www.sintef.no
36. Institute for Energy Technology (IFE) homepage: www.ife.no
37. Ole B. Rygg, John D. Friedeman and Jan Nossen, "Advanced Well Flow Model Used for Production, Drilling and Well Control Applications", Presented at the IADC Well Control Conference for Europe, Aberdeen, May 22-24 1996.
38. Product information taken from SPT Groups websites, available from: http://www.sptgroup.com/en/Products/olga/OLGA-ABC/ - Visited 23.02.11
39. Ø. Arild, K.K Fjelde, A. Merlo, ENI-E&P Div., B. Daireaux, T. Løberg, "Blowflow – a New Tool Within Blowout Risk Management", IADC/SPE 114568 presented at IADC/SPE Asia Pacific Drilling Technology Conference and Exhibition, Jakarta, Indonesia, 25-27 August 2008.
40. Kjetil Aleksander Moe, "BlowFlow – a Software Tool Developed by IRIS for Risk Based Evaluation of Blowout Scenarios" - Bachelor Thesis, Department of Petroleum Engineering, University of Stavanger, February 2008.
41. IRIS homepage, available from: www.iris.no
42. Samuel F. Noynaert and Jerome J. Schubert, "Modeling Ultra-Deepwater Blowouts and Dynamic Kills and the Resulting Blowouts Control Best Practice Recommendations" - SPE/IADC 92626, Presented at the SPE/IADC Drilling Conference, Amsterdam, The Netherlands, 23-25 February 2005.

43. Samuel F. Noynaert, “Ultra Deepwater Blowouts: COMASim Dynamic Kill Simulator Validation and Best Practice Recommendations”, Master Thesis, Texas A&M University, December 2004 – Available from: http://repository.tamu.edu/bitstream/handle/1969/1543/etd-tamu-2004C-PETE-Noynaert.pdf?sequence=1 – Downloaded 23.02.11
44. Halliburton homepage, information available from: http://www.halliburton.com/ps/default.aspx?pageid=898&navid=923
45. Scanpower`s homepage, information available from: http://www.scandpower.com/products/blow-fam/
46. Helge J. Dervo, Bjarne Blom-Jensen, “Comparison of Quantitative Blowout Risk Assessment Approaches” - SPE 86706, March 2004.
47. Brochure taken from SPT Group`s website, available from: http://www.sptgroup.com/upload/documents/Brochures/drillbench5.pdf
48. Conversations with Professor Kjell Kåre Fjelde at the University of Stavanger during the time period from 01.03.2011 - 15.04.2011.
49. I.Tiselj and S. Petelin, “Modelling of Two-Phase Flow with Second-Order Accurate Scheme”, Journal of Computational Physics, 136, p. 503-521, Article no: CP975778, - 1997.
50. Carlos Osornio V., Humberto Castro Mtz., Victor Vallejo A., Enrique Ayala V., “Successful Well Control in the Cantarell Field Applying the Dynamic Method” – SPE 71372, Presented at the 2001 SPE Annual Technical Conference and Exhibition, New Orleans, Louisiana, - 2001.
51. T.Ølberg, T. Gilhuus, F. Leraand, and J. Haga, “Re-Entry and Relief Well Drilling To Kill an Underground Blowout in a Subsea Well: A Case History of Well 2/4-14” - SPE/IADC 21991, Presented at the SPE/IADC Drilling Conference in Amsterdam, - 1991.
52. Curtis F. Gerald, Patrick O. Wheatley, “Applied Numerical Analysis” (Fourth Edition), California Polytechnic State University San Luis Obispo, ISBN 0-201-52825-8., Addison-Wesley Publishing Company - 1989.

Appendices

Appendix A – Two-Phase Code for Oil and Liquid (16 pages)

Appendix B – Results (17 pages)

Appendix A – Two-Phase Code for Oil and Liquid

A1.

```
% Tranisent two-phase code based on AUSMV scheme: Oil and Water
% Note: All variables with subscripts g (for gas), means oil in this
% code.

clear;

% Geometry data for blowout well:
welldepth = 4000; % [m]
nobox = 50;
nofluxes = nobox+1;
dx = welldepth/nobox;
%dt = 0.005;
dt= 0.005;
dtdx = dt/dx;
time = 0.0;
endtime = 1300;
nosteps = endtime/dt;
timebetweensavingtimedata = 5;
nostepsbeforesavingtimedata = timebetweensavingtimedata/dt;

% Slip parameters of gas:
k = 1.0;
s = 0.0;

% Viscosities (Pa*s):
viscl = 0.001; % Liquid (water)
viscg = 0.002; % Oil

% Density parameters:
% Liquid density at stc and speed of sound in liquid:
  dstc = 1000.0; % [kg/m^3]
  pstc = 100000.0; % [Pa]
  al = 1500;
  t1 = dstc-pstc/(al*al);
% Ideal gas law constant
  rt = 100000;

% Gravity acceleration

  grav = 9.81;

% BOP Well status (open = 1, closed = 0)

  wellopen = 1.0;
  wellclosed = 0.0;
  opening = wellopen;

% Choke status:
  choke = 0;



% Define and intilalize flow variables
```

```
% Check area[m^2]:

   for i = 1:nobox
    do(i)=0.2167; %[m]
    di(i)=0.127;  %[m]

    areal(i) = 3.14/4*(do(i)*do(i)- di(i)*di(i));
    arear(i) = 3.14/4*(do(i)*do(i)- di(i)*di(i));
    area(i) = 3.14/4*(do(i)*do(i)- di(i)*di(i));
    ang(i)=3.14/2;

   end

   temp = arear(2);

   for i = 1:2
     do(i) = 0.216;
     di(i) = 0.168;
     areal(i) = 3.14/4*(do(i)*do(i)- di(i)*di(i));
     arear(i) = 3.14/4*(do(i)*do(i)- di(i)*di(i));
   end
     arear(2)= temp;

   arear(nobox) = arear(nobox-1);

    for i = 1:nobox
        % Density of liquid and gas[kg/m^3]:
        dl(i) = 1000.0;
        dg(i)= 1.0;
        %"Old" density is set equal to new density to calculate new values
        %based on the old ones:
        dlo(i)= dl(i);
        dgo(i)=dg(i);
        % Velocity of liquid and gas at new and previous timesteps[m/s]:
        vl(i) = 0.0;
        vlo(i)= 0.0;
        vg(i)= 0.0;
        vgo(i)= 0.0;
        %Pressure at "top", the pressure in the horisontal pipe is the same
        %all over:
        p(i) = 100000.0;
        po(i) = p(i);
        %Phase volume fractions of gas and liquid:
        eg(i)= 0.0;
        ego(i)=eg(i);
        ev(i)=1-eg(i);
        evo(i)=ev(i);

        vg(i)=0.0;
        vgo(i)=0.0;
        vl(i)=0.0;
        vlo(i)=0.0;

        % Variables related to the velocity of the flux boundaries at old
        %and new times, and on the left and right side of the boxes:
        vgr(i)=0.0;
        vgor(i)= 0.0;
        vgl(i)= 0.0;
        vgol(i)= 0.0;
```

```
         vlr(i)=0.0;
         vlor(i)=0.0;
         vll(i)=0.0;
         vlol(i)=0.0;

   % Conservative variables:

        qv(i,1)=dl(i)*ev(i)*(areal(i)+arear(i))*0.5;
        qvo(i,1)=qv(i,1);

        qv(i,2)=dg(i)*eg(i)*(areal(i)+arear(i))*0.5;
        qvo(i,2)=qv(i,2);

        qv(i,3)=(qv(i,1)*vl(i)+qv(i,2)*vg(i))*(areal(i)+arear(i))*0.5;
        qvo(i,3)=qv(i,3);

    end

% Intialize fluxes:

for i = 1:nofluxes
  for j =1:3
   flc(i,j)=0.0; % Flux of liquid over box boundary
   fgc(i,j)=0.0; % Flux of gas over box boundary
   fp(i,j)= 0.0; % Pressure flux over box boundary
  end
end


%  Main program. Here we will progress in time. First som intializations
% and definitions to take out results:

countsteps = 0;
counter=0;
printcounter = 1;
pbot(printcounter) = p(1);
pchoke(printcounter)= p(nobox);
pcasingshoe(printcounter)=p(25);
liquidmassrateout(printcounter) = 0;
gasmassrateout(printcounter)=0;
timeplot(printcounter)=time;

for i = 1:nosteps
   countsteps=countsteps+1;
   counter=counter+1;
   time = time+dt;

% First of all a dirty trick is used in order to make the well vertical.
% The pipe was initialised for a horisontal case. However, for a vertical
% case we would need a steady state solver. The gravity constant g is
adjusted from
% zero to 9.81 m/s2 during 100 seconds (corresponds to hoisting the well
% from a horisontal postion to vertical case:

       if (time <= 100)
         g = grav*time/100;
       else
         g = grav;
```

```
       end



% Checking the boundary conditions:
% Here we specify the inlet rates of the different phases at the
% bottom of the pipe in [kg/s]. We interpolate to make things smooth.


if (time < 150)

  inletligmassrate=0.0;
  inletgasmassrate=0.0;
  Pres = 0.0;

elseif ((time>=150) & (time < 170))
  inletligmassrate = 0.0; %7.5*(time-150)/20;
  inletgasmassrate=0.0;
  Pres = 50000000; %The reservoir pressure is set to a constant value.

elseif (time >= 170) & (time < 800)
  inletligmassrate = 0.0;  %PI model starts.
  Pres = 50000000;

elseif (time >= 800) & (time < 900)
  inletligmassrate = 190.0*(time-800)/100; % Required dynamic kill rate
  Pres = 50000000;

 elseif (time >= 900)
   inletligmassrate = 190.0; % Required dynamic kill rate
   Pres = 50000000;
 end


% Specify the outlet pressure:
 pressureoutlet = 100000.0;

% Based on these boundary values combined with use of extrapolations
techniques
% for the remaining unknowns at the boundaries, we will define the mass and
% momentum fluxes at the boundaries (inlet and outlet of pipe).

% Inlet fluxes first:

     flc(1,1)= inletligmassrate/areal(1);
     flc(1,2)= 0.0;
     flc(1,3)= flc(1,1)*vlo(1);

     fgc(1,1)= 0.0;
     fgc(1,2)= inletgasmassrate/areal(1);
     fgc(1,3)= fgc(1,2)*vgo(1);

     fp(1,1)= 0.0;
     fp(1,2)= 0.0;
     fp(1,3)= po(1)+0.5*(po(1)-po(2));

%     end
```

```matlab
% Outlet fluxes (open & closed conditions):

    if (opening>0.01)
        flc(nofluxes,1)= dlo(nobox)*evo(nobox)*vlo(nobox);
        flc(nofluxes,2)= 0.0;
        flc(nofluxes,3)= flc(nofluxes,1)*vlo(nobox);

        fgc(nofluxes,1)= 0.0;
        fgc(nofluxes,2)= dgo(nobox)*ego(nobox)*vgo(nobox);
        fgc(nofluxes,3)= fgc(nofluxes,2)*vgo(nobox);

        fp(nofluxes,1)= 0.0;
        fp(nofluxes,2)= 0.0;
        fp(nofluxes,3)= pressureoutlet;
    else

        if (choke == 0.0)
         flc(nofluxes,1)= 0.0;
         flc(nofluxes,2)= 0.0;
         flc(nofluxes,3)= 0.0;

         fgc(nofluxes,1)= 0.0;
         fgc(nofluxes,2)= 0.0;
         fgc(nofluxes,3)= 0.0;

         fp(nofluxes,1)=0.0;
         fp(nofluxes,2)=0.0;
         fp(nofluxes,3)= po(nobox)-0.5*(po(nobox-1)-po(nobox));

        else

         flc(nofluxes,1)= dlo(nobox)*evo(nobox)*vlo(nobox);
         flc(nofluxes,2)= 0.0;
         flc(nofluxes,3)= flc(nofluxes,1)*vlo(nobox);

         fgc(nofluxes,1)= 0.0;
         fgc(nofluxes,2)= dgo(nobox)*ego(nobox)*vgo(nobox);
         fgc(nofluxes,3)= fgc(nofluxes,2)*vgo(nobox);

         fp(nofluxes,1)= 0.0;
         fp(nofluxes,2)= 0.0;
         fp(nofluxes,3)= pressureoutlet;


        end

    end

% Now we will find the fluxes between the different cells:

     for j = 2:nofluxes-1
      cl = csound(ego(j-1),po(j-1),dlo(j-1),k);
      cr = csound(ego(j),po(j),dlo(j),k);
      c = max(cl,cr);
      pll = psip(vlor(j-1),c,evo(j));
      plr = psim(vlol(j),c,evo(j-1));
      pgl = psip(vgor(j-1),c,ego(j));
      pgr = psim(vgol(j),c,ego(j-1));
```

```
     vmixr = vlol(j)*evo(j)+vgol(j)*ego(j);
     vmixl = vlor(j-1)*evo(j-1)+vgor(j-1)*ego(j-1);

     pl = pp(vmixl,c);
     pr = pm(vmixr,c);
     mll= evo(j-1)*dlo(j-1);
     mlr= evo(j)*dlo(j);
     mgl= ego(j-1)*dgo(j-1);
     mgr= ego(j)*dgo(j);

     flc(j,1)= mll*pll+mlr*plr;
     flc(j,2)= 0.0;
     flc(j,3)= mll*pll*vlor(j-1)+mlr*plr*vlol(j);

     fgc(j,1)=0.0;
     fgc(j,2)= mgl*pgl+mgr*pgr;
     fgc(j,3)= mgl*pgl*vgor(j-1)+mgr*pgr*vgol(j);

     fp(j,1)= 0.0;
     fp(j,2)= 0.0;
     fp(j,3)= pl*po(j-1)+pr*po(j);
    end

% Fluxes have now been calculated. We will now update the conservative
% variables in each of the numerical cells. First liquid mass, then gas
% mass and finally momemtum equation.

     sum = 0.0;
    for j=1:nobox
     vmixfric = vlo(j)*evo(j)+vgo(j)*ego(j);
     viscmix =  viscl*evo(j)+viscg*ego(j);
     densmix = dlo(j)*evo(j)+dgo(j)*ego(j);
     a2 = arear(j);
     a1 = areal(j);
     avg = (a2+a1)*0.5;


     pressure=p(j);
     pressure3=p(3);
     pressure50=p(nobox);



     friclossgrad(j) =
dpfric(vlo(j),vgo(j),evo(j),ego(j),dlo(j),dgo(j),pressure,do(j),di(j),viscl
,viscg);

     sum = sum +(friclossgrad(j)*(welldepth/nobox)); % Friction loss in
                                                     % the blowout well
[Pa]


    PIvalue = 0.000001;


     if j==1
         PIreal= 0.0;
     elseif j==2
```

```
            PIreal = 0.0;
        elseif (j==3)
              if (time < 170)
                    PIreal = 0.0;
               elseif ((time >= 170) & (time < 600))
                         PIreal = PIvalue * (time-170)/430;
               elseif (time >= 600)
                        PIreal = PIvalue;
              end

        elseif j>3
               PIreal = 0.0;
        end

       if (Pres>pressure3)
%          PImodel = PIreal*(Pres-pressure3)*(800+(pressure50-
100000.0)/(1100*1100));
          PImodel = PIreal*(Pres-pressure3)*800;
       else
          PImodel = 0.0;
       end

        if (j==3)
         PItest=PImodel;
        end

        qv(j,1)=qvo(j,1)-dtdx*((a2*flc(j+1,1)-a1*flc(j,1))...
                              +(a2*fgc(j+1,1)-a1*fgc(j,1))...
                              +(avg*fp(j+1,1)-avg*fp(j,1)));

        qv(j,2)=qvo(j,2)-dtdx*((a2*flc(j+1,2)-a1*flc(j,2))...
                              +(a2*fgc(j+1,2)-a1*fgc(j,2))...
                              +(avg*fp(j+1,2)-avg*fp(j,2)-PImodel));
                              %PImodel is added for oil

        qv(j,3)=qvo(j,3)-dtdx*((a2*flc(j+1,3)-a1*flc(j,3))...
                              +(a2*fgc(j+1,3)-a1*fgc(j,3))...
                              +(avg*fp(j+1,3)-avg*fp(j,3)))...
                     -dt*avg*((friclossgrad(j))+g*densmix);

       end

       %Simple friction model:
       %(32*vmixfric*viscmix/(do(j)*do(j))+g*densmix);

       %More complex model:
       %friclossgrad
       %Function of the model:
       %((2*f*densmix*vmixfric*abs(vmixfric))/(do(j)-di(j))+g*densmix);



% Section where we find the physical variables (pressures, densities etc)
% from the conservative variables. Some trickes to ensure stability


       for j=1:nobox
```

```
      qv(j,1)= qv(j,1)/(areal(j)+arear(j))*2.0;
      qv(j,2)= qv(j,2)/(areal(j)+arear(j))*2.0;

      if (qv(j,1)<0.00000001)
        qv(j,1)=0.0;
      end

      if (qv(j,2)< 0.00000001)
        qv(j,2)=0.0000001;
      end


% The primitive variables can be found direct from the conservative
% variables with the use of the analytical solution.
% To find the primitive variables on a more flexible way, to easier
% convert the code to oil and water, a numerical solution can be used
% instead:


      % Coefficients:
      a = 1/(al*al);
      b = t1-qv(j,1)-rt*qv(j,2)/(al*al);
      c = -1.0*t1*rt*qv(j,2);

      % Analytical solution:
      %p(j)=(-b+sqrt(b*b-4*a*c))/(2*a);   % Pressure
      %dl(j)= dstc + (p(j)-pstc)/(al*al); % Density of liquid
      %dg(j) = p(j)/rt;                   % Density of gas

      %Numerical Solution:
      [p(j),error]=itsolver(po(j),qv(j,1),qv(j,2)); % Pressure
      dl(j)=rholiq(p(j)); % Density of liquid
      dg(j)=rogas(p(j)); % Density of gas

      % Incase a numerical solution is not found, the program will write
out "error":
      if error > 0
         error
      end

      eg(j)= qv(j,2)/dg(j);
      ev(j)=1-eg(j);

 %Plotting of f(p)
%  pressure = p(j);
%  dl(j)=rholiq(p(j)); % Density of liquid
%  dg(j)=rogas(p(j)); % Density of gas
%
%  w1=dl(j)*(1-ego(j));
%  w2=dg(j)*ego(j)
%
%  for i = 1:50
%     f(p(j))= w1-(dl(j)*(1-(w2/dg(j))));
%  end


% The conservative variables:

      qv(j,1)=qv(j,1)*(areal(j)+arear(j))/2.0;
```

```
      qv(j,2)=qv(j,2)*(areal(j)+arear(j))/2.0;

%      vg(j)=qv(j,3)/(dl(j)*ev(j)+dg(j)*eg(j));
%      vl(j)=vg(j);

% Part where we interpolate in the slip parameters to avoid a
% singularities. In the transition to one-phase gas flow, we need to
% have a smooth transition to no-slip conditions.

      xint = (eg(j)-0.75)/0.25;
      k0 = k;
      s0 = s;
      if ((eg(j)>=0.75) & (eg(j)<=1.0))
        k0 =1.0*xint+k*(1-xint);
        s0 = 0.0*xint+s*(1-xint);
      end

      if (eg(j)>=0.999999)
        k1 = 1.0;
        s1 = 0.0;
      else
        k1 = (1-k0*eg(j))/(1-eg(j));
        s1 = -1.0*s0*eg(j)/(1-eg(j));
      end
 %     help1 = dl(j)*ev(j)*k1+dg(j)*eg(j)*k0;
 %     help2 = dl(j)*ev(j)*s1+dg(j)*eg(j)*s0;

 %     vmixhelp = (qv(j,3)-help2)/help1;
 %     vg(j)=k0*vmixhelp+s0;
 %     vl(j)=k1*vmixhelp+s1;
      help1 = qv(j,3)/(dl(j)*ev(j)+dg(j)*eg(j));

      vll(j)= help1/areal(j);
      vlr(j)= help1/arear(j);
      vgl(j)= vll(j);
      vgr(j)= vlr(j);


 %    Test slip parameters & areachange:

      help1 = dl(j)*ev(j)*k1+dg(j)*eg(j)*k0;
      help2 = dl(j)*ev(j)*s1+dg(j)*eg(j)*s0;

      vmixhelpl = (qv(j,3)/areal(j)-help2)/help1;
      vgl(j)=k0*vmixhelpl+s0;
      vll(j)=k1*vmixhelpl+s1;

      vmixhelpr = (qv(j,3)/arear(j)-help2)/help1;
      vgr(j)=k0*vmixhelpr+s0;
      vlr(j)=k1*vmixhelpr+s1;

 %  Averaging velocities:

      vl(j)= 0.5*(vll(j)+vlr(j));
      vg(j)= 0.5*(vgl(j)+vgr(j));

    end
```

```matlab
% Old values are now set equal to new values in order to prepare
% computation of next time level.
    for j = 1:nobox
     po(j)=p(j);
     dlo(j)=dl(j);
     dgo(j)=dg(j);
     vlo(j)=vl(j);
     vgo(j)=vg(j);
     ego(j)=eg(j);
     evo(j)=ev(j);

     vlor(j)=vlr(j);
     vlol(j)=vll(j);
     vgor(j)=vgr(j);
     vgol(j)=vgl(j);

      for m =1:3
       qvo(j,m)=qv(j,m);
      end
    end


% Section where we save some timedependent variables in arrays.
% e.g. the bottomhole pressure. They will be saved for certain
% timeintervalls defined in the start of the program in order to ensure
% that the arrays do not get to long!

  if (counter>=nostepsbeforesavingtimedata)
    printcounter=printcounter+1;
    time
    pbot(printcounter)= p(1);
    pi(printcounter)= PItest;
    pchoke(printcounter)=p(nobox);
    pcasingshoe(printcounter)=p(25);
%
liquidmassrateout(printcounter)=dl(nobox)*ev(nobox)*vl(nobox)*area(nobox);
%
gasmassrateout(printcounter)=dg(nobox)*eg(nobox)*vg(nobox)*area(nobox);

liquidmassrateout(printcounter)=dl(nobox)*ev(nobox)*vl(nobox)*arear(nobox);

gasmassrateout(printcounter)=dg(nobox)*eg(nobox)*vg(nobox)*arear(nobox);
    timeplot(printcounter)=time;
    counter = 0;


  end
end

% End of stepping forward in time.


% Relief well data:

%Assuming that the relief well is filled with water.
dstc = 1000; %[kg/m^3]
viscl = 0.001; %[Pa*s]
vline = 20.21; % [m/s]
```

```
vr = 7.85; % [m/s]
vdp = 7.95;% [m/s]
vdc = 14.07;% [m/s]
doline = 0.0762; % Choke and kill line ID=3"
dor = 0.2167; % ID = 8,53" of 9 5/8" casing.
dir = 0.127; % OD = 5" of drillpipe = ID in cased section.
dooh = 0.2159; % Open hole 8,5"
didp = 0.127; % Drill Pipe 5"
didc = 0.1714; % Drill collar 6 3/4 "

%Measured depth:
lck = 100; % Both the choke and the kill line is 100 m long
lr = 5500; % MD of cased section
ldp = 350; % MD of open hole section with DP
ldc = 150; % MD of open hole section with DC


%Reynolds number for choke/kill lines, relief well and openhole section:
Reline = (dstc*vline*(doline))/viscl;
Rer = (dstc*vr*(dor-dir))/viscl;
Redp = (dstc*vdp*(dooh-didp))/viscl;
Redc = (dstc*vdc*(dooh-didc))/viscl;

%Friction pressure drop given in [kg/m^2*s^2]:

%For choke and kill line:

if Reline < 0
    dpdxline = 0;
elseif Reline <= 2000
   dpdxline = (4/(doline))*(16/Reline)*0.5*dstc*(vline*vline);
elseif Reline >= 4000
      dpdxline = (4/(doline))*(0.046*(Reline)^-0.2)*0.5*dstc*(vline*vline);
end

% For cased part of the relief well:

if Rer <= 2000
   dpdxr = (4/(dor-didp))*(16/Rer)*0.5*dstc*(vr*vr);
elseif Rer >= 4000
      dpdxr = (4/(dor-didp))*(0.046*(Rer)^(-0.2))*0.5*dstc*(vr*vr);
end

%For open hole section and DP:

if Redp <= 2000
   dpdxdp = (4/(dooh-didc))*(16/Redp)*0.5*dstc*(vdp*vdp);
elseif Redp >= 4000
      dpdxdp = (4/(dooh-didc))*(0.046*(Redp)^(-0.2))*0.5*dstc*(vdp*vdp);
end

%For open hole section and DC:

if Redc <= 2000
   dpdxdc = (4/(dooh-didc))*(16/Redc)*0.5*dstc*(vdc*vdc);
elseif Redc >= 4000
      dpdxdc = (4/(dooh-didc))*(0.046*(Redc)^(-0.2))*0.5*dstc*(vdc*vdc);
end
```

```
%Total friction gradient in relief well given in [Pa]:
totfric = ((dpdxline)*lck*2) + (dpdxr*lr) + ((dpdxdp*ldp)+ (dpdxdc*ldc));
% The friction factor for kill and choke lines are multilied with 2,
% to find the friction factor in both lines.

% Pumping pressure down relief well [Pa]:
Ppump=totfric + sum;


% The blowing well is killed when DynKill > Pres:

DynKill = p(3);
pressure = p(j);

if DynKill>=Pres
   'Dynamic Kill is successful'
elseif DynKill < Pres
         'The well is not killed'
end


% Printing of resultssection

countsteps

plot(timeplot,pbot/100000)
%plot(timeplot,pchoke/100000)
%plot(timeplot,pcasingshoe/100000)
%plot(timeplot,liquidmassrateout)
%plot(timeplot,gasmassrateout)
%plot(timeplot,pi)
%plot(vg)
%plot(friclossgrad*80/100000)
%plot(eg)
%plot(ev)
%plot(dg)
%plot(dl)
```

A2.

```
function mixsoundvelocity = csound(gvo,po,dlo,k)
% Note that at this time k is set to 1.0 (should maybe be
% included below

temp= gvo*dlo*(1.0-gvo);
a=1;
if (temp < 0.01)
  temp = 0.01;
end

cexpr = sqrt(po/temp);

%if (gvo <= 0.5)
 %mixsoundvelocity = min(cexpr,1000);
%else
% mixsoundvelocity = min(cexpr,316);

 mixsoundvelocity=1500.0*(1-gvo)+1100*gvo;
end
```

A3.

```
function friclossgrad =
dpfric(vlo,vgo,evo,ego,dlo,dgo,pressure,do,di,viscl,viscg)

%friclossgrad =
%dpfric(vlo,vgo,evo,ego,dlo,dgo,pressure,do,di,viscl,viscg)
% Works for two phase flow. The one phase flow model is used but mixture
 % values are introduced.

 rhol = rholiq(pressure);
 rhog = rogas(pressure);
 vmixfric = vlo*evo+vgo*ego;
 viscmix =  viscl*evo+viscg*ego;
 densmix = dlo*evo+dgo*ego;

 % Calculate mix reynolds number
 Re = ((densmix*abs(vmixfric)*(do-di))/viscmix);

 % Calculate friction factor. For Re > 3000, the flow is turbulent.
 % For Re < 2000, the flow is laminar. Interpolate in between.

 if (Re<0.001)
     f=0.0;
 else
    if (Re >= 3000)
    f = 0.052*Re^(-0.19);
    elseif ( (Re<3000) & (Re > 2000))
    f1 = 24/Re;
    f2 = 0.052*Re^(-0.19);
    xint = (Re-2000)/1000.0;
    f = (1.0-xint)*f1+xint*f2;
    else
    f = 24/Re;
```

```
    end
 end

 friclossgrad = ((2*f*densmix*vmixfric*abs(vmixfric))/(do-di));


end
```

A4.

```
function [press,error] = itsolver(p,qv1,qv2)

% The numerical solver implementeted here for solving the equation f(x)= 0
% "wellpressure(p)= 0" is called the
% Method of Halving the Interval (Bisection Method)

% You will not find exact match for f(x)= 0. Maybe f(x) = 0.0001. By using
% ftol we say that if f(x)<ftol, we are satisfied. Since our function
% gives results in Pascal, we say that ftol = 1000 Pa gives us a quite good
% answer.

 ftol = 0.00001;
 ftol = 0.001;

 % Specify the search interval". xguess is the pressure you guess for the
 % pressure. (Remember x is in Pa). 1 Bar = 100 000 Pa.

 % Set number of iterations to zero

  noit = 0;
  error = 1.0; % Error is set to zero because we havent any input
information yet.

  i = 0;
  while (error > 0)
      i = i+1;
      xguess = p;
      xint = 150000*i;
      x1 = xguess-xint/2.0;
      x2 = xguess+xint/2.0;

 f1 = qv1-rholiq(x1)*(1.0-qv2/rogas(x1));
 f2 = qv1-rholiq(x2)*(1.0-qv2/rogas(x2));

 % First include a check on whether f1xf2<0. If not you must adjust your
 % initial search intervall. If error is 1 and zero pbot, then you must
 % adjust the intervall here.


 if (f1*f2)>=0
     error = 0;
     if (f1<ftol)
         press=x1;
     elseif (f2<ftol)
         press=x2;
     else
```

```
         error=1;
     end

 else
 % start iterating, we are now on the track.
     x3 = (x1+x2)/2.0;
     f3 = qv1-rholiq(x3)*(1.0-qv2/rogas(x3));

     while (f3>ftol | f3 < -ftol)
        noit = noit +1 ;

        if (f3*f1) < 0
           x2 = x3;
        else
           x1 = x3;
        end

        x3 = (x1+x2)/2.0;
        f3 = qv1-rholiq(x3)*(1.0-qv2/rogas(x3));
        f1 = qv1-rholiq(x1)*(1.0-qv2/rogas(x1));

     end
     error = 0;
     press = x3;
     noit;
 end
  end
```

A5.

```
function pmvalue = pm(v,c)


  if (abs(v)<=c)
    pmvalue = -1.0*(v-c)*(v-c)/(4*c)*(-2.0-v/c)/c;
  else
    pmvalue = 0.5*(v-abs(v))/v;
  end
end
```

A6.

```
function pmvalue = pp(v,c)

  if (abs(v)<=c)
    pmvalue = (v+c)*(v+c)/(4*c)*(2.0-v/c)/c;
  else
    pmvalue = 0.5*(v+abs(v))/v;
  end
end
```

A7.

```
function pmvalue = psim(v,c,alpha)

  if (abs(v)<=c)
    pmvalue = -1.0*alpha*(v-c)*(v-c)/(4*c)+(1-alpha)*(v-abs(v))/2;
  else
    pmvalue = 0.5*(v-abs(v));
  end
end
```

A8.

```
function pmvalue = psip(v,c,alpha)


  if (abs(v)<=c)
    pmvalue = alpha*(v+c)*(v+c)/(4*c)+(1-alpha)*(v+abs(v))/2;
  else
    pmvalue = 0.5*(v+abs(v));
  end
end
```

A9.

```
function [rhol] = rholiq(pressure)
%Simple model for liquid density
p0 = 100000.0; % Assumed

rhol = 1000.0 + (pressure-p0)/(1500.0*1500.0);
end
```

A10.

```
function rhog = rogas(pressure)

%Simple gas density model. Temperature is neglected.
% rhogas = pressure / (velocity of sound in the gas phase)^2 = pressure /
% rT -->

  %rhog = pressure/100000.0;

  %For oil:

   rhog = 800+(pressure-100000.0)/(1100*1100);
```

Appendix B – Results

B1 - Verification of the Stability

Scenario: A 1000 meter deep well with inflow of gas (q_{gas}) and liquid (q_{liq}) in the bottom of the well. The well is assumed to have ID = 3.5” and OD = 6.3”. The viscosities for gas and liquid is assumed to be $\mu_g = 0{,}005\ cP$ and $\mu_{liq} = 50\ cP$, and the slip parameters is set to $k = 1{,}2$ and $s = 0{,}5$ (Ref. eq. 3.7). Flow rates of gas and liquid is set to $0{,}15\ kg/$s and $7{,}5\ kg/s$, unless otherwise stated.

Note, that the friction model given in eq. 3.10 - Section 3.3 is used in these simulations.

Reducing the Number of Time Steps

- Analytical approach:

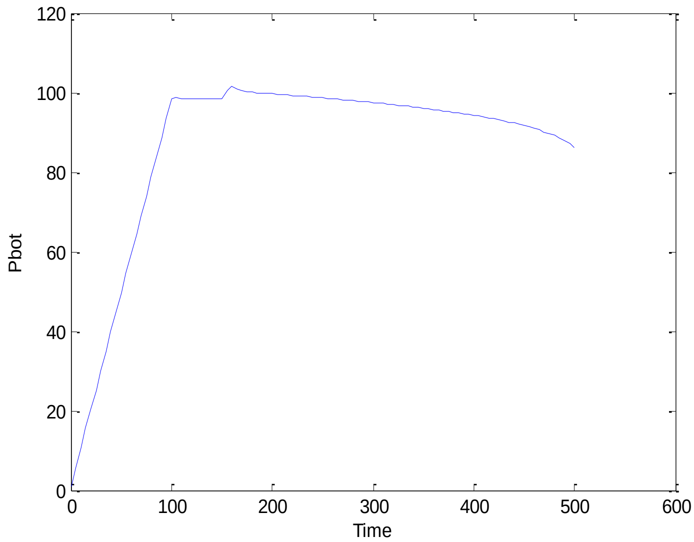


Figure B1-1: Bottom hole pressure [bar] vs. Time [s], Time step = 0,005 seconds

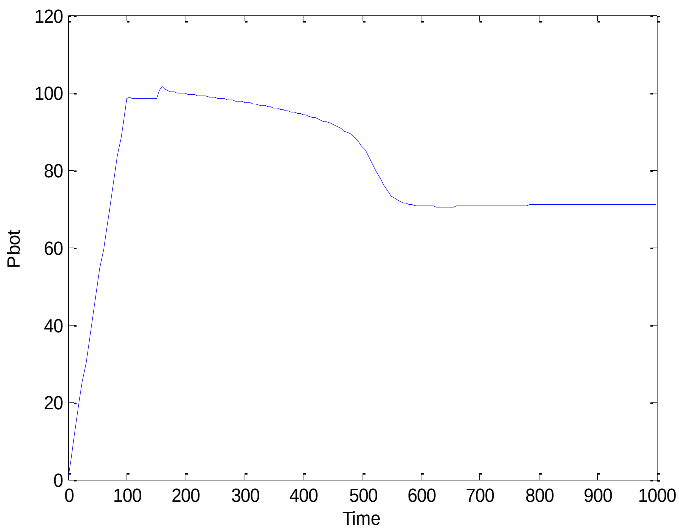


Figure B1-2: Bottom hole pressure [bar] vs. Time [s], Time step = 0,002 seconds

- Numerical approach:

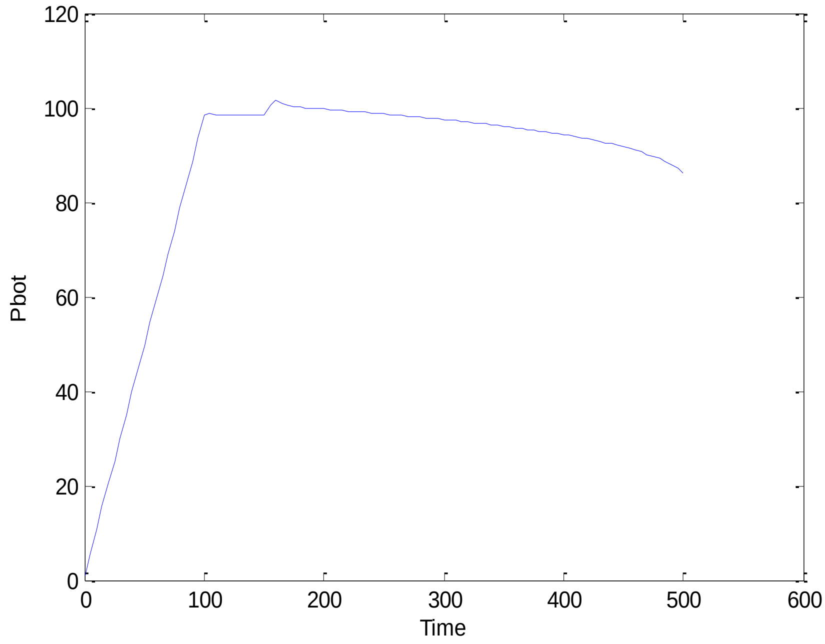


Figure B1-3: Bottom hole pressure [bar] vs. Time [s], Time step = 0,005 seconds

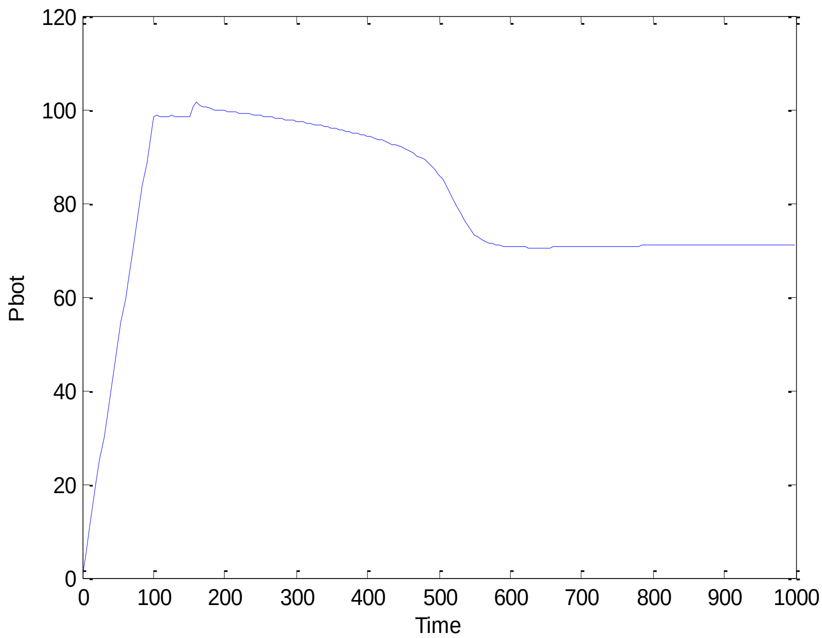


Figure B1-4: Bottom hole pressure [bar] vs. Time [s], Time step = 0,002 seconds

Changing Inflow Rates

- Analytical approach:

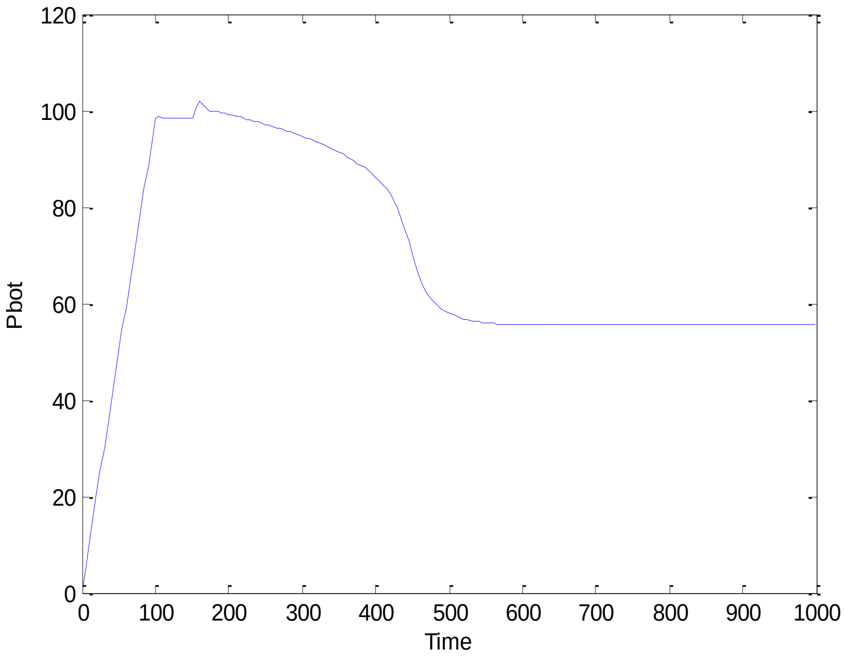


Figure B1-5: Bottom hole pressure [bar] vs. Time [s], For $q_{gas} = 0{,}30\ kg/s$ and $q_{liq} = 7{,}5\ kg/s$

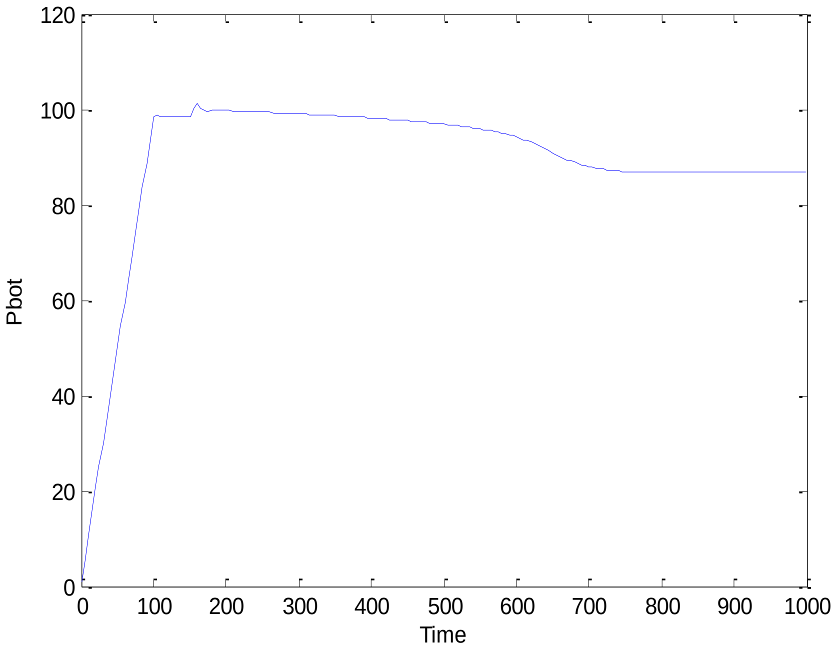


Figure B1-6: Bottom hole pressure [bar] vs. Time [s], For $q_{gas} = 0{,}05\ kg/s$ and $q_{liq} = 6{,}5\ kg/s$

- Numerical approach:

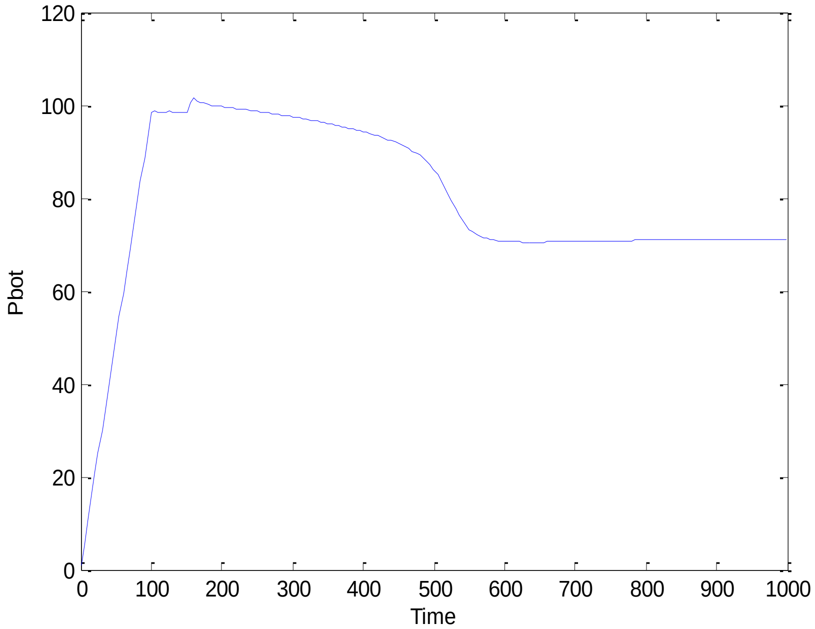


Figure B1-7: Bottom hole pressure [bar] vs. Time [s], For $q_{gas} = 0{,}15\ kg/s$ and $q_{liq} = 7{,}5\ kg/s$

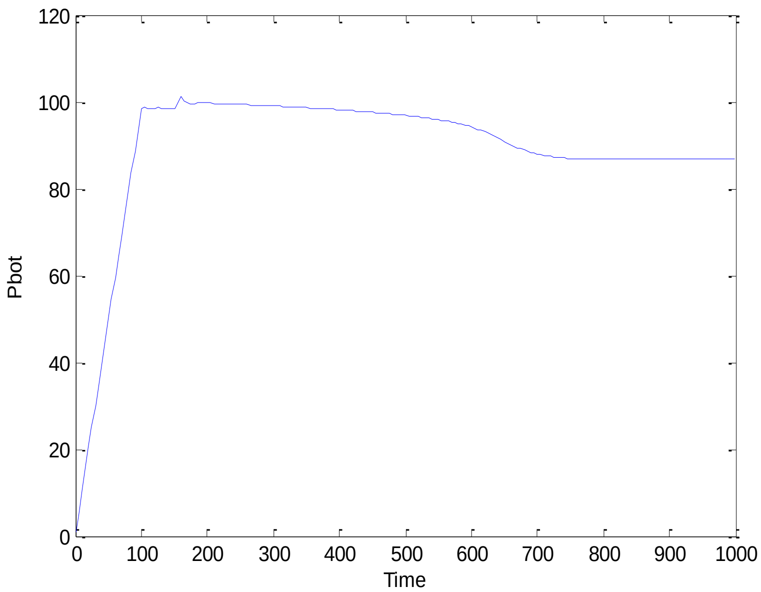


Figure B1-8: Bottom hole pressure [bar] vs. Time [s], For $q_{gas} = 0{,}05\ kg/s$ and $q_{liq} = 6{,}5\ kg/s$

B2 - Results from First Simulation

The scenario modeled is described in detail in section 5.2.1 and 5.2.3. The simulations are performed with the following parameters:

- Velocities of sound for oil and water: $316\ m/s$ and $1000\ m/s$, respectively.
- Viscosities of oil and water: $\mu_o = 2\ cP,\ \mu_l = 1\ cP$
- Slip parameters: $k = 1{,}0$, $s = 0$ (no-slip assumption)

Note: In all simulations the PI model is implemented in box number three.

For Dynamic Kill Rate: $q_{l,in} = 100\ kg/s$

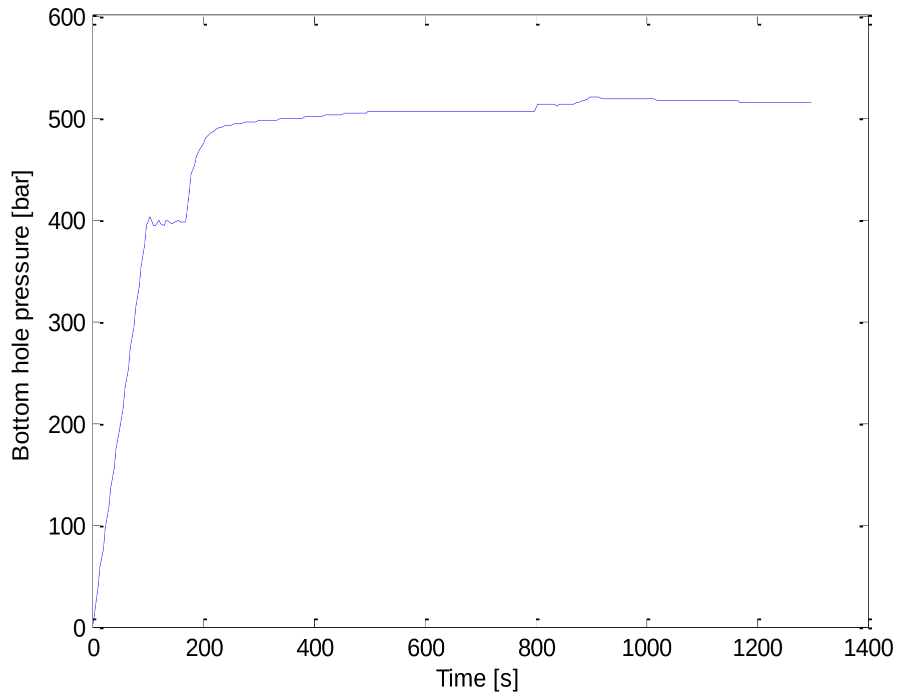


Figure B2-1: BHP $[bar]$ vs. Time $[s]$, Dynamic Kill Rate: $q_{l,in} = 100\ kg/s$

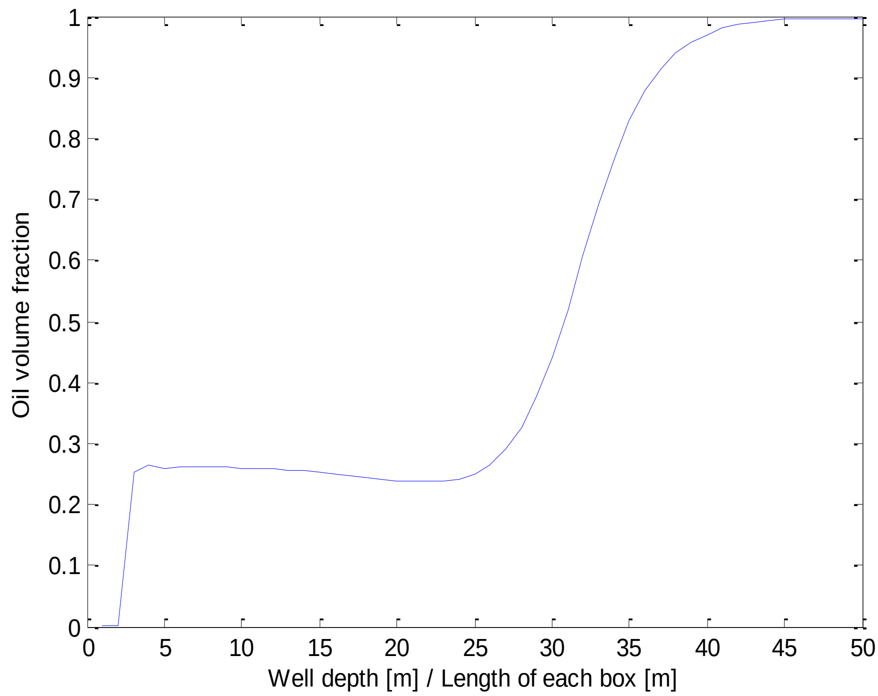


Figure B2-2: Oil Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 100\ kg/s$

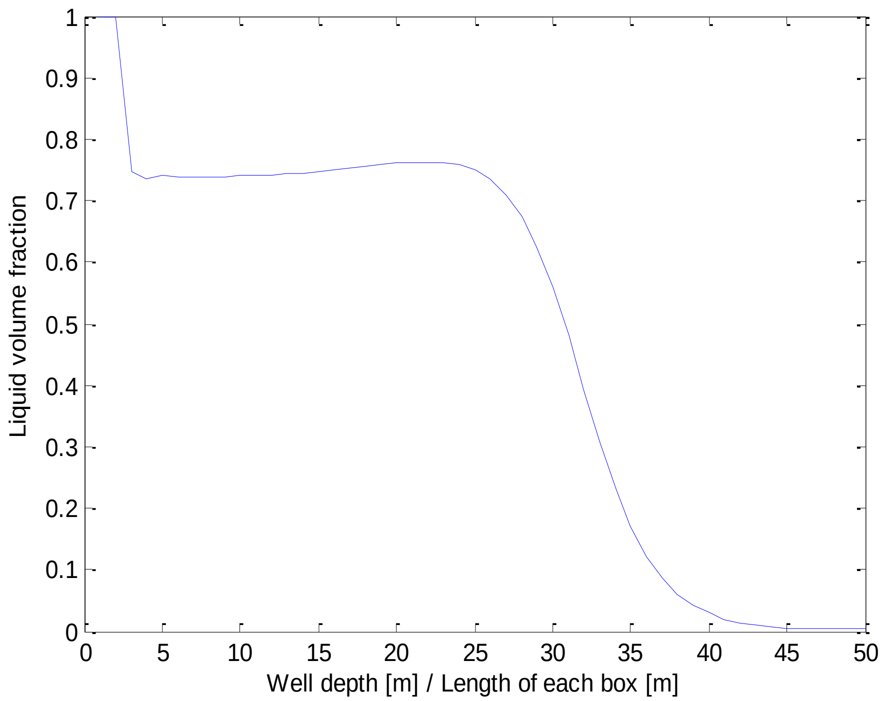


Figure B2-3: Liquid Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 100\ kg/s$

For Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

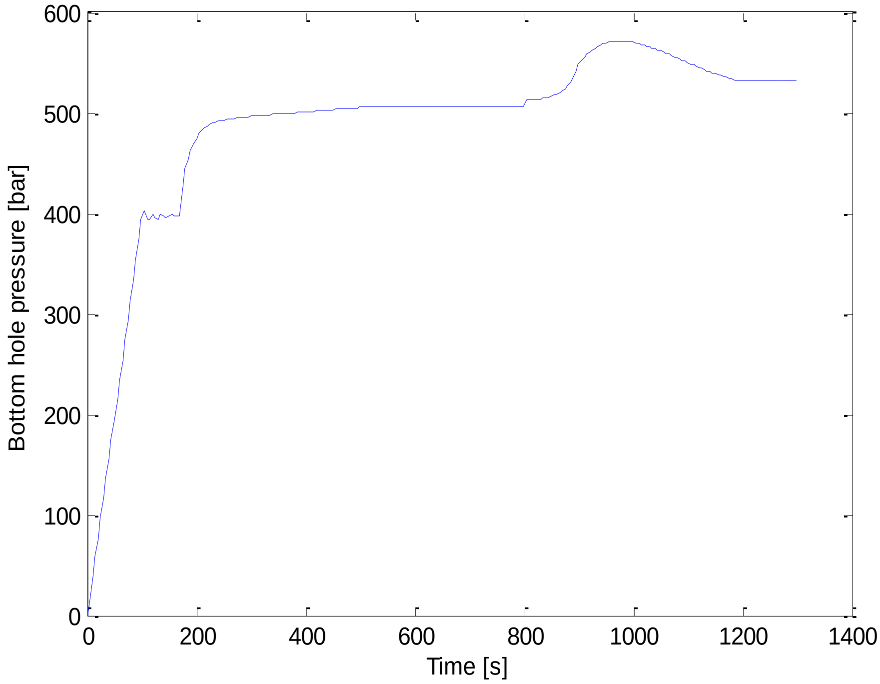


Figure B2-4: BHP $[bar]$ vs. Time $[s]$, Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

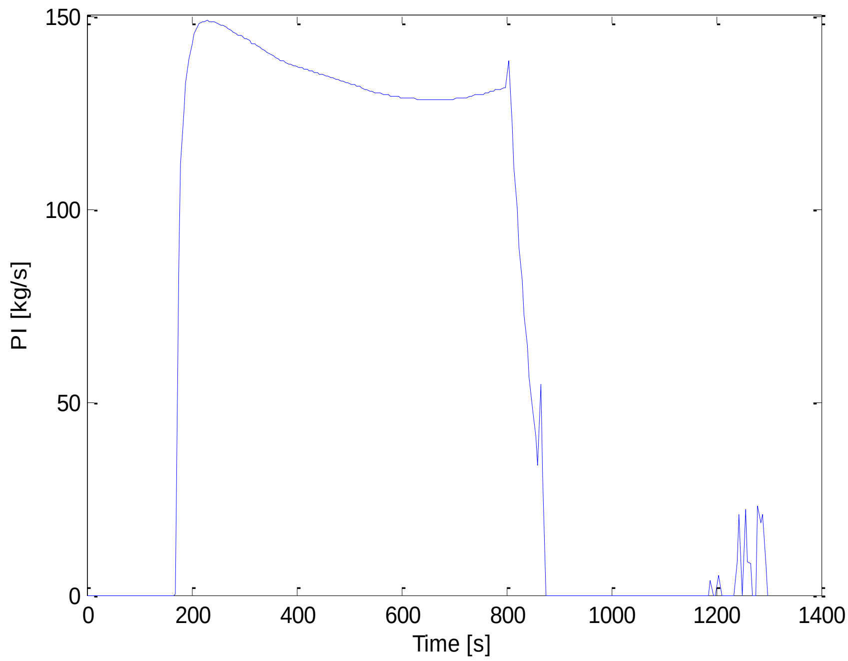


Figure B2-5: PI $[kg/s]$ vs. Time $[s]$, Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

<u>For Dynamic Kill Rate: $q_{l,in} \ = \ 160 \ kg/s$</u>

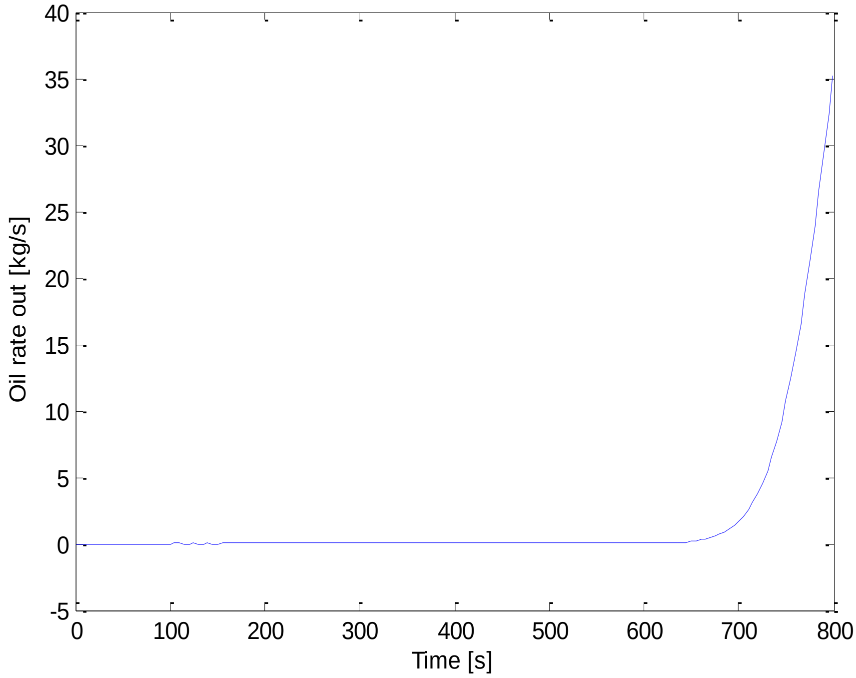


Figure B2-6: Oil Rate Out $[kg/s]$ vs. Time $[m]$, Dynamic Kill Rate: $q_{l,in} \ = \ 160 \ kg/s$

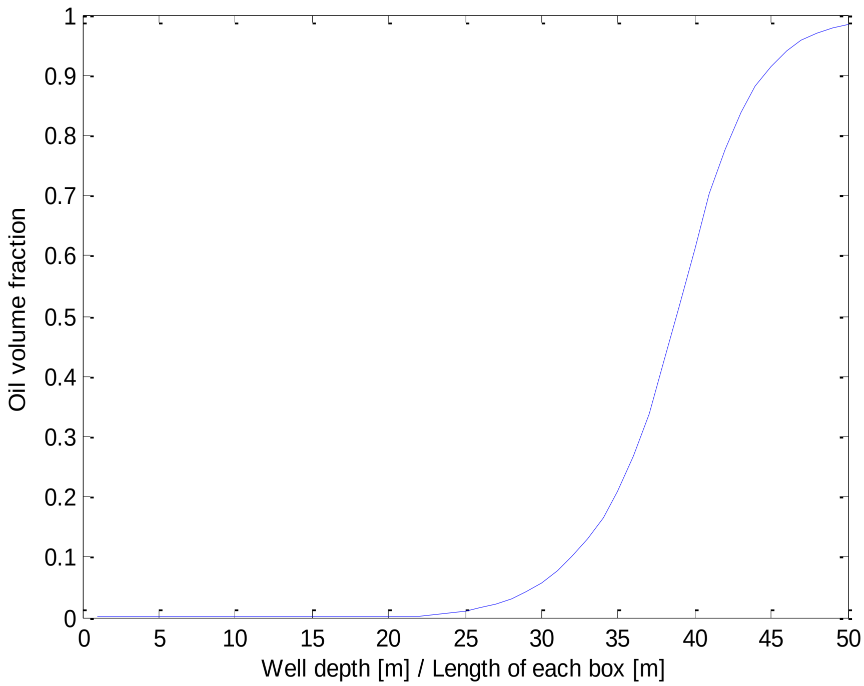


Figure B2-7: Oil Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} \ = \ 160 \ kg/s$

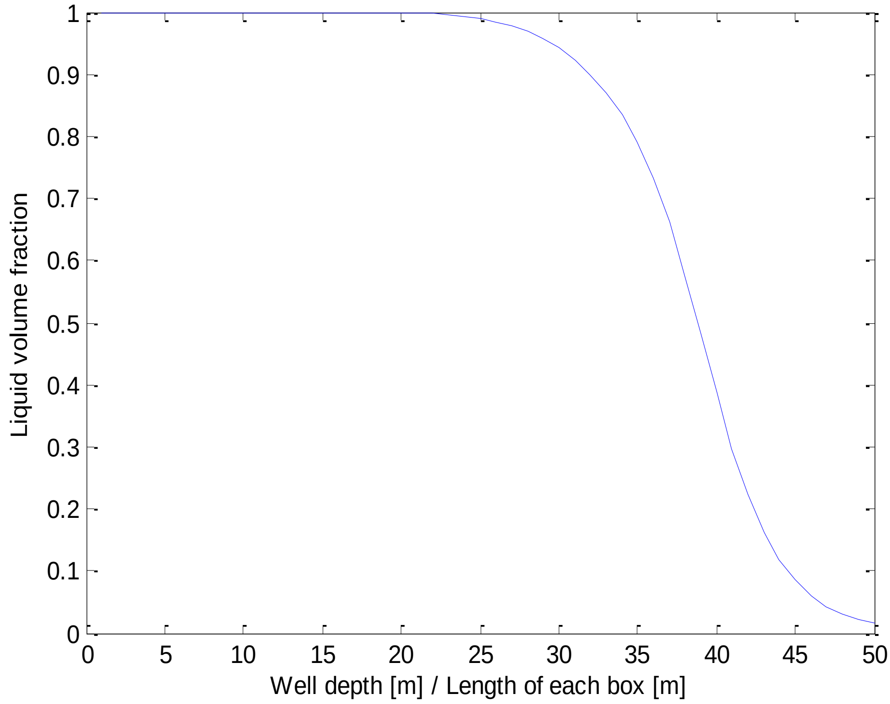


Figure B2-8: Liquid Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 160\ kg/s$

B3 - Results from Second Simulation

The scenario modeled is described in detail in section 5.2.1 and 5.2.4. The simulations are performed with the following parameters:

- Velocities of sound in oil and water: $1100\ m/s$, $1500\ m/s$, respectively [31].
- Viscosities of oil and water: $\mu_o = 2\ cP, \mu_l = 1\ cP$
- Slip parameters: $k = 1{,}0$, $s = 0$ (no-slip assumption)

Note: In all simulations the PI model is implemented in box number three, unless stated otherwise.

For Dynamic Kill Rate: $q_{l,in} = 90\ kg/s$

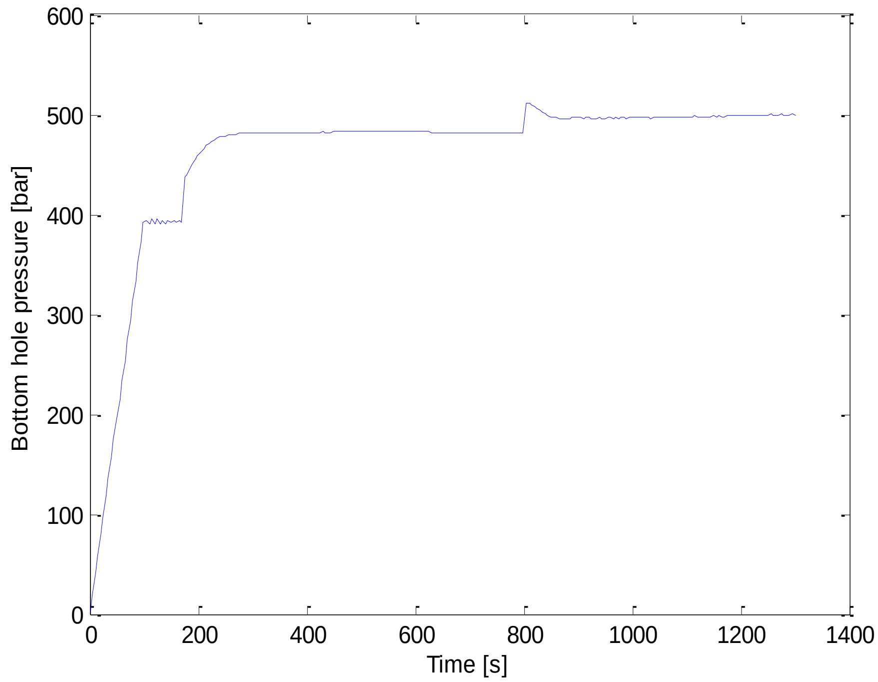


Figure B3-1: BHP $[bar]$ vs. Time $[m]$, Dynamic Kill Rate: $q_{l,in} = 90\ kg/s$

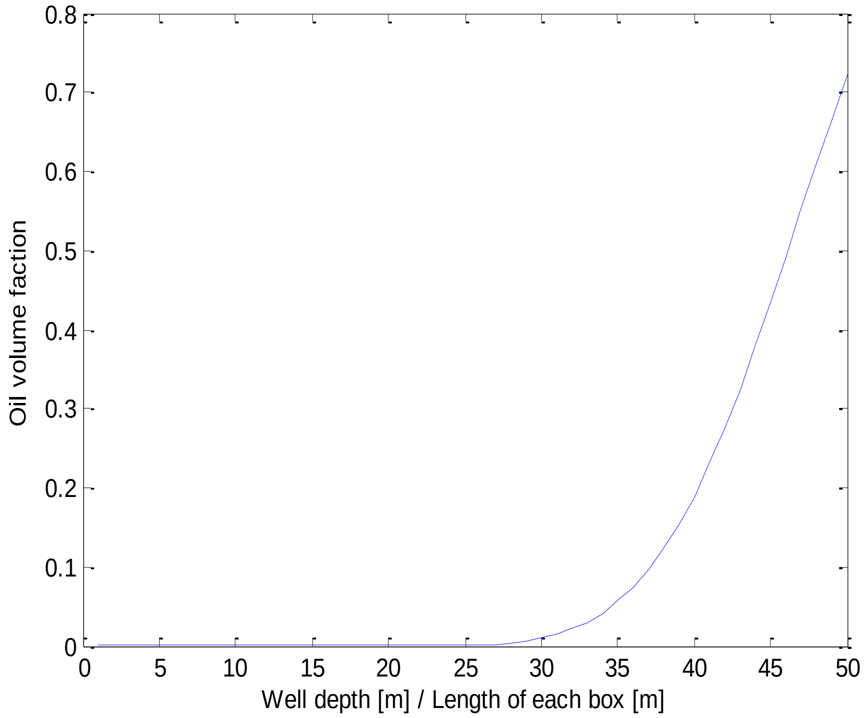


Figure B3-2: Oil Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 90\ kg/s$

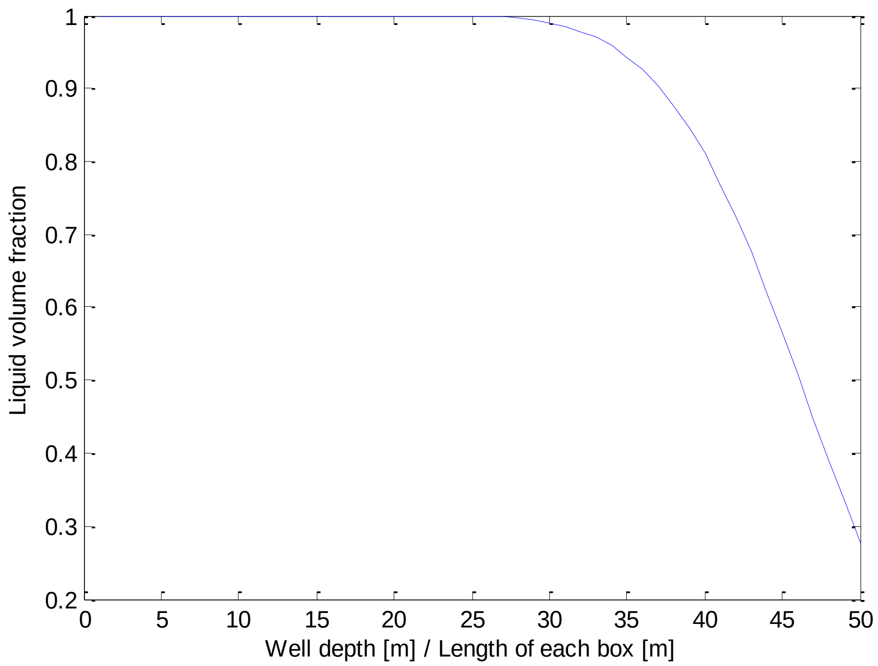


Figure B3-3: Liquid Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 90\ kg/s$

For Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

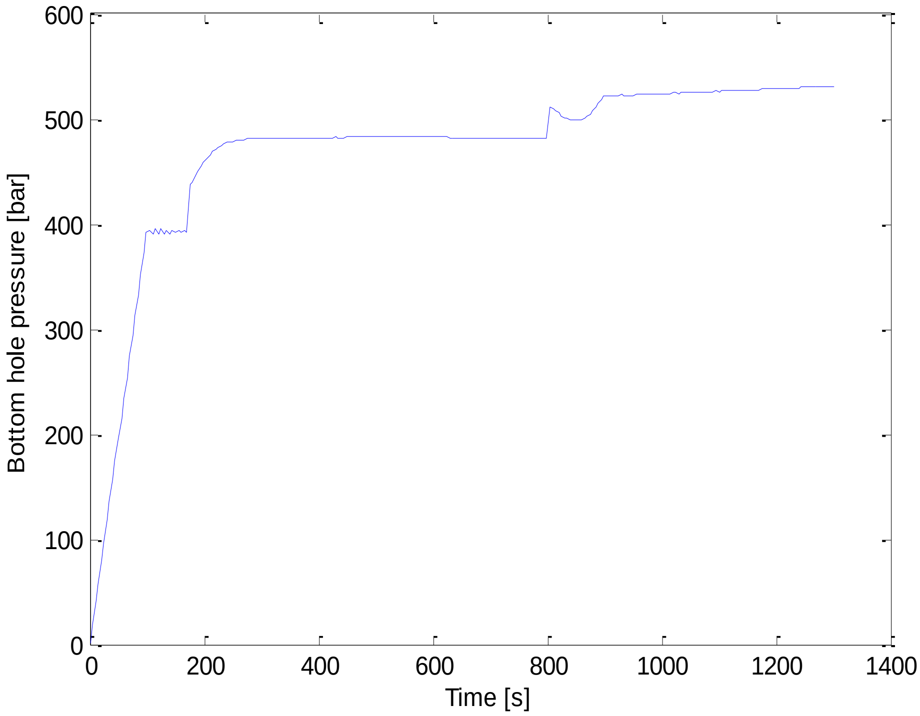


Figure B3-4: BHP $[bar]$ vs. Time $[m]$, Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

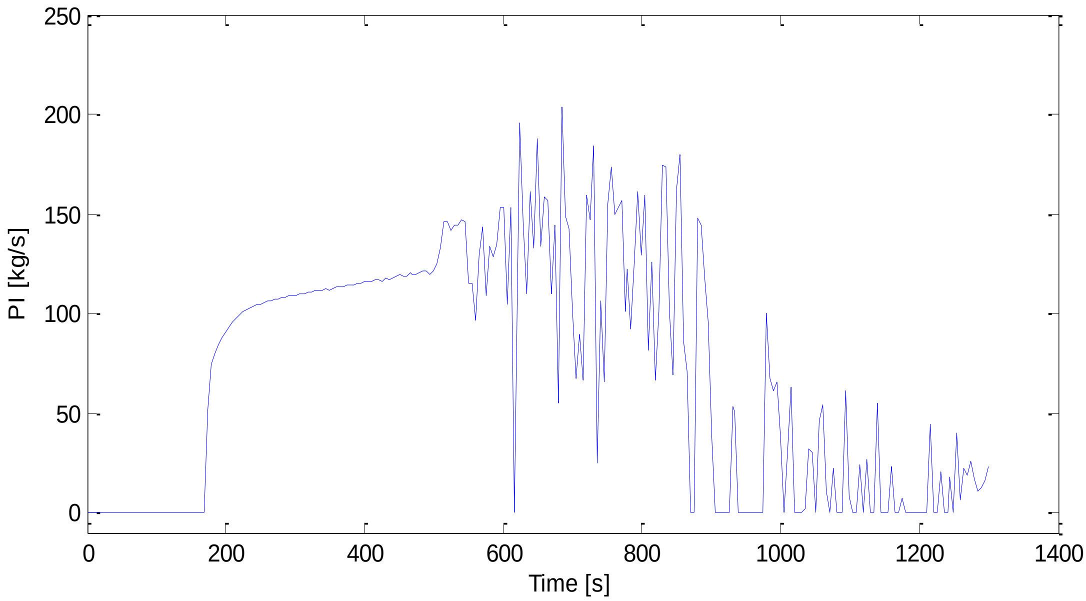


Figure B3-5: PI $[kg/s]$ vs. Time $[m]$, Dynamic Kill Rate: $q_{l,in} \ = \ 150 \ kg/s$

For Dynamic Kill Rate: $q_{l,in} = 190\ kg/s$ - Part 1

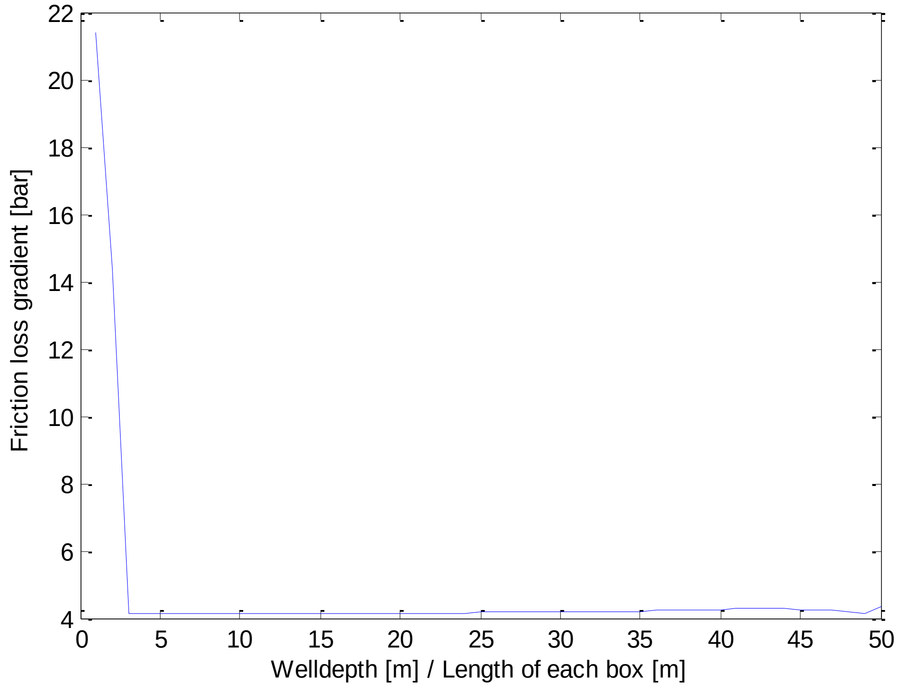


Figure B3-6: Friction Loss Gradient $[bar]$ vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 190\ kg/s$

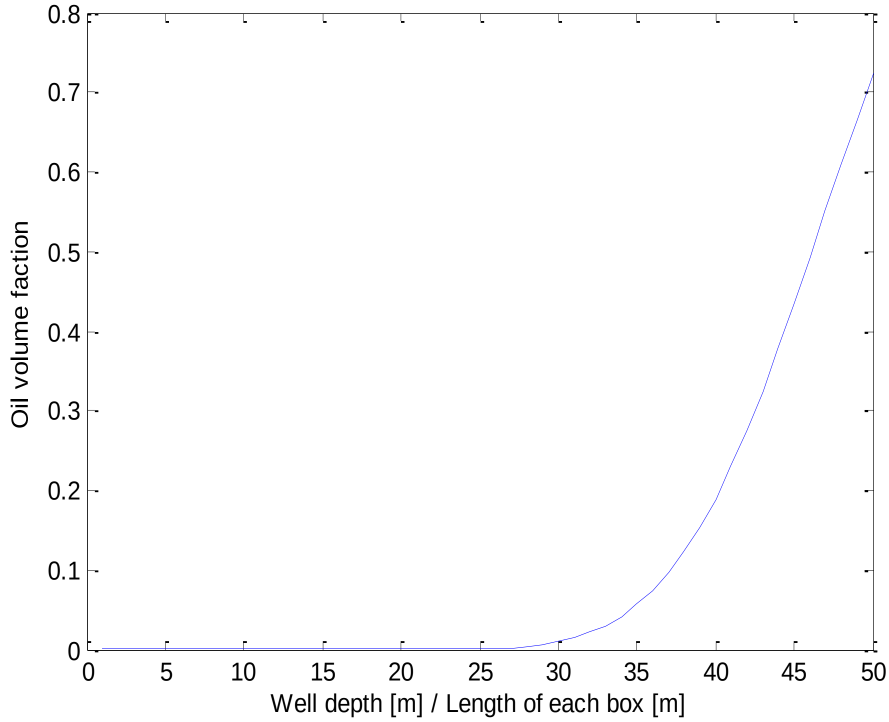


Figure B3-7: Oil Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 190\ kg/s$

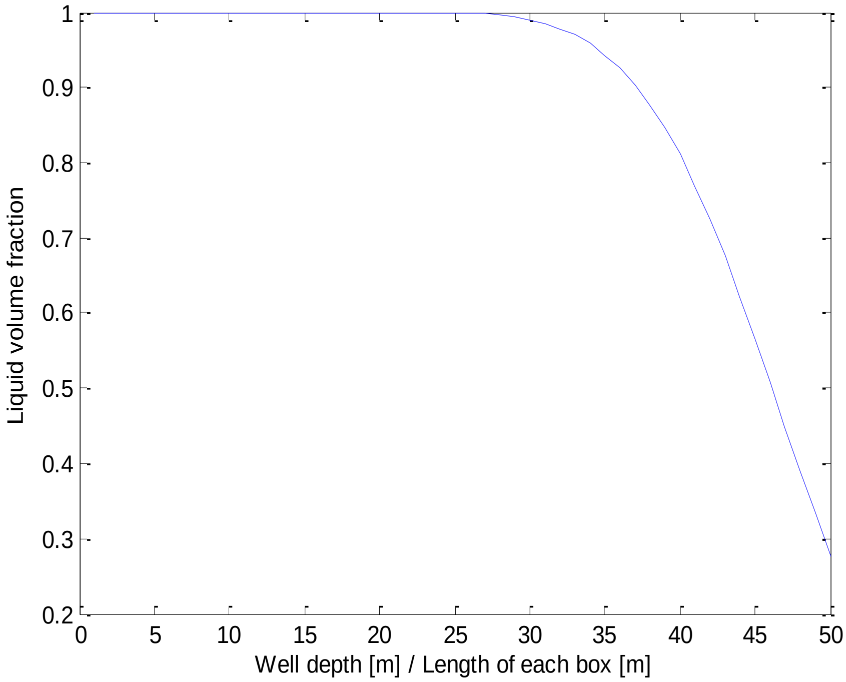


Figure B3-8: Liquid Volume Fraction vs. Well Depth $[m]$/Length of Each Box $[m]$, Dynamic Kill Rate: $q_{l,in} = 190\ kg/s$

For Dynamic Kill Rate: $q_{l,in} = 190\ kg/s$ - Part 2

The following simulation have been run with a time step of $0{,}0005$ seconds and the PI model implemented in box number two, to verify that the PI model was acting as a stiff source term requiring integration through a time step far below the CFL-criterion.

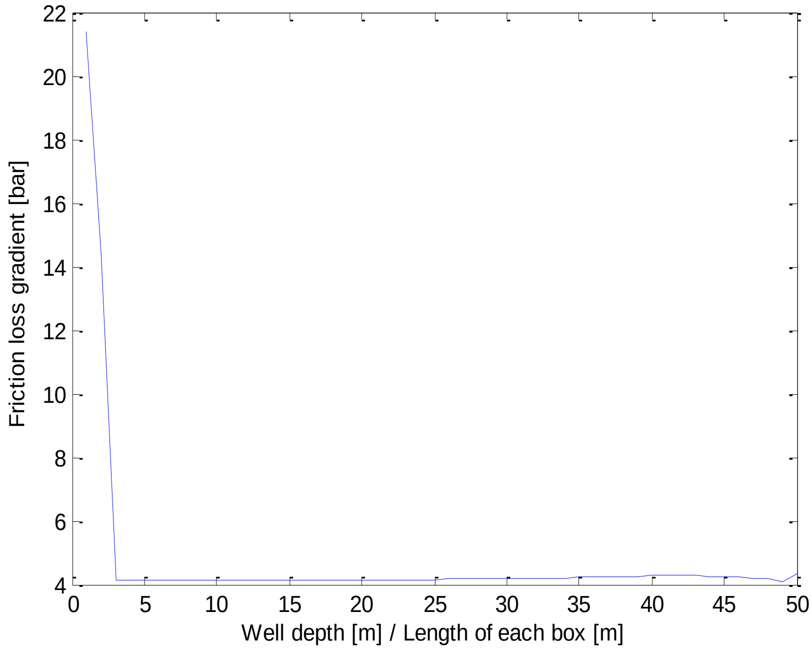


Figure B3-9: Friction Loss Gradient [bar] vs. Well Depth [m]/Length of Each Box [m], Dynamic Kill Rate: $190\ kg/s$

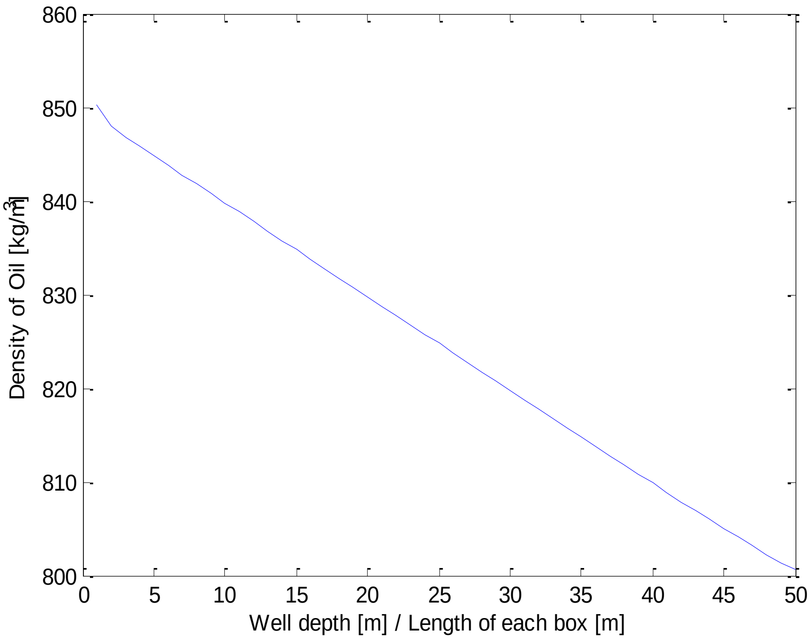


Figure B3-10: Density of Oil [kg/m^3], Dynamic Kill Rate: $190\ kg/s$

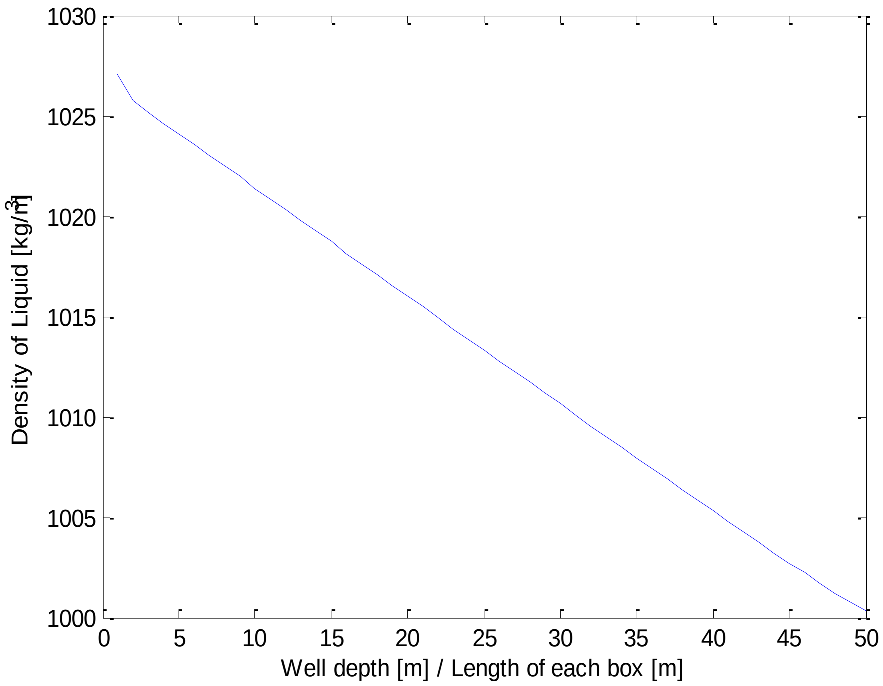


Figure B3-11: Density of Liquid [kg/m^3], Dynamic Kill Rate: $190\ kg/s$

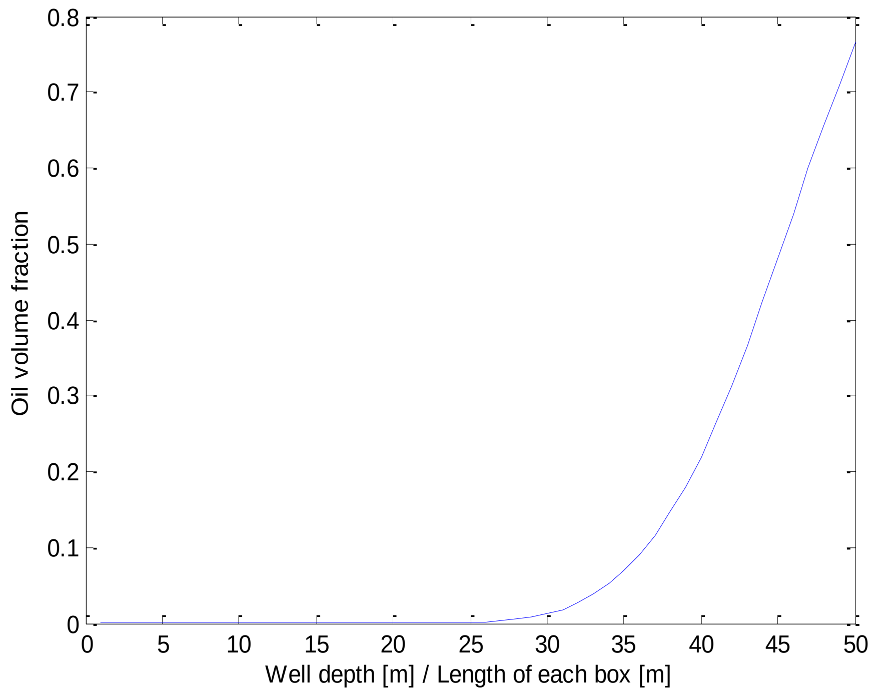


Figure B3-12: Oil Volume Fraction vs. Well Depth [m]/Length of Each Box [m], Dynamic Kill Rate: $190\ kg/s$

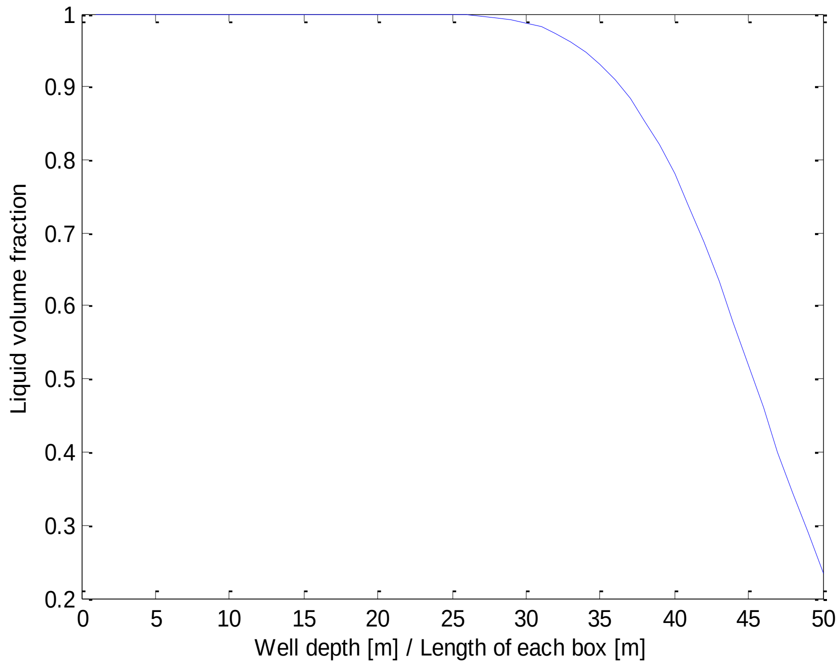


Figure B3-13: Liquid Volume Fraction vs. Well Depth [m]/Length of Each Box [m], Dynamic Kill Rate: $190\ kg/s$