EQUIVALENT CIRCULATING DENSITY CONTRIBUTION
TO THE PLASTERING EFFECT OF CASING WHILE DRILLING TECHNOLOGY:
ANALYSIS OF ANNULAR FLUID VELOCITY AND ANNULAR PRESSURE THROUGH
COMPUTATIONAL FLUID DYNAMICS

by
Comert Satkan
A thesis submitted to the Faculty and the Board of Trustees of the Colorado School of Mines in partial fulfillment of the requirements for the degree of Master of Science (Petroleum Engineering).

Golden, Colorado
Date ______________
Signed: ______________________
Comert Satkan

Signed: ______________________
Dr. Alfred William (Bill) Eustes III
Thesis Advisor

Golden, Colorado
Date ______________
Signed: ______________________
Dr. William Fleckenstein
Professor and Head
Department of Petroleum Engineering
ABSTRACT

Application of Casing while Drilling (CwD) technology has accelerated in the recent decade. The technology came up with so many benefits and unknowns. The plastering effect of CwD and specific hydraulics conditions constituted the main portion of the research on this technology. The plastering effect has been shown to be beneficial in many aspects and there is an interest to define governing factors on it. In an effort to describe the physics beyond the plastering effect, several components that potentially govern the process are under investigation.

This thesis is a theoretical modeling study; and, it is focused on one of the potential conditions for the plastering effect of CwD. Annulus hydraulics is elaborated to investigate the pressure and velocity profiles. While doing so, Computational Fluid Dynamics (CFD) has been used in the form provided by ANSYS, Inc. Fluent commercial software package. The main focus is on the responses of annular fluid velocity and annular pressure to varying geometry with increasing eccentricity. Furthermore, a combination of rotational motion of the inner pipe and eccentricity is studied. Multiple physical explanations of the flow field in diverse conditions are described. Visuals in the form of contour plots and X-Y plots verified physical explanations are presented. The discussions and interpreted results are given in the interpretation of CFD results chapter.

The knowledge presented in the results section and the computational models, especially detailed information presented about annular pressure distribution in axial and horizontal plane, is expected to assist in furthering studies as the link between the high equivalent circulating density of CwD and plastering effect is of interest.

In summary, hydraulics for the CwD technology is empirically found to be unique, and this can be a major contributor to the plastering effect, which provides significant achievements to the CwD technology.
# TABLE OF CONTENTS

ABSTRACT ........................................................................................................................... iii

TABLE OF CONTENTS ........................................................................................................ iv

LIST OF FIGURES ............................................................................................................... vii

LIST OF TABLES ................................................................................................................ xi

ACKNOWLEDGEMENT ........................................................................................................ xii

CHAPTER 1 INTRODUCTION ............................................................................................... 1

1.1 Motivation of Study ...................................................................................................... 3

1.2 Objectives .................................................................................................................... 4

1.3 Thesis Outline ............................................................................................................. 4

CHAPTER 2 LITERATURE REVIEW .................................................................................... 6

2.1 Casing while Drilling Technology .............................................................................. 6

2.1.1 Casing while Drilling Process .............................................................................. 7

2.1.2 Advantages of Casing while Drilling Technology ............................................... 11

2.1.2.1 Enhanced well economics and timing ......................................................... 11

2.1.2.2 Elimination of swab and surge pressure effects ........................................... 13

2.1.2.3 Rig adaptations and HSE ............................................................................ 14

2.1.2.4 Wellbores in gauge ....................................................................................... 14

2.1.2.5 Improvements on production ........................................................................ 15

2.1.2.6 Wellbore strengthening and lost circulation reduction ................................ 16

2.1.3 Limitations of Casing while Drilling Technology ............................................... 21

2.2 Review of Hydraulics ............................................................................................... 24
2.2.1 Drilling Fluid Rheology ................................................................. 24
2.2.2 Fluid Flow and Frictional Pressure Loss Analysis for a CwD annulus .......... 27

CHAPTER 3 .................................................................................................. 32
METHODOLOGY ......................................................................................... 32
3.1 Application of Computational Fluid Dynamics in Drilling Engineering ............. 32
3.2 ANSYS – Fluent CFD Package .................................................................. 33

3.2.1 Pre-Process Stage ............................................................................... 34
  3.2.1.1 Creation of geometry ................................................................. 35
  3.2.1.2 Mesh generation ...................................................................... 35
  3.2.1.3 Material properties ................................................................. 39
  3.2.1.4 Boundary conditions ............................................................... 40
3.2.2 CFD Solver ....................................................................................... 41
  3.2.2.1 Solution initialization ............................................................... 41
  3.2.2.2 Convergence monitoring ......................................................... 42
3.2.3 Post-Processing ................................................................................ 42
3.3 CFD Model Design .............................................................................. 43
  3.3.1 Geometry of the CFD Model ......................................................... 43
  3.3.2 Meshing of the CFD Model ........................................................... 47
  3.3.3 Assumptions and Boundary Conditions ........................................ 54
  3.3.4 Model Validation ......................................................................... 56

INTERPRETATION OF CFD RESULTS ..................................................... 60
4.1 Effect of Casing Eccentricity in Stationary Pipe Conditions ............................ 62
4.1.1 Concentric Annulus With Laminar Flow and Without Rotation .................................................66
4.1.2 30% Eccentric Annulus With Laminar Flow and Without Rotation .................................67
4.1.3 60% Eccentric Annulus With Laminar Flow and Without Rotation ...........................................68
4.2 Effect of Casing Rotation and Eccentricity ........................................................................77
4.2.1 Concentric Annulus With Laminar Flow and With Rotation (80rpm) ...................................78
4.2.2 30% Eccentric Annulus With Laminar Flow and With Rotation (80rpm) .....................79
4.2.3 60% Eccentric Annulus With Laminar Flow and With Rotation (80rpm) .....................80

CHAPTER 5 ........................................................................................................................................91
CONCLUSIONS, DISCUSSION AND FUTURE WORK .................................................................91
5.1 Discussion and Future Work ........................................................................................................92

REFERENCES ....................................................................................................................................94

APPENDIX A. FIGURES SHOWING EFFECT OF ECCENTRICITY AND INNER PIPE ROTATION ON ANNULAR PRESSURE AND VELOCITY PROFILES ......................................................99
LIST OF FIGURES

Figure 1.1: Eccentric flow in annulus..................................................................................................................5
Figure 2.1: (a) Non-retrievable CwD assembly, and (b) Conventional drilling assembly.................................7
Figure 2.2: Casing Drive System mounted to top drive........................................................................................9
Figure 2.3: Drillable PDC bit....................................................................................................................................9
Figure 2.4: Retrievable CwD Bottom Hole Assembly set for directional drilling.............................................10
Figure 2.5: Improved drilling curve of Akamba -2 well. Solid line represents actual operation and dashed line represents planned operation.................................................................12
Figure 2.6: Effectively modified operational mud weight window after removing trip margins .......................13
Figure 2.7: Borehole quality improvement by CwD (right) as compared to conventional drilling (left) ............15
Figure 2.8: Contact angle of CwD (right) is smaller than contact angle of drill pipe (left). \( \alpha_1 > \alpha_2 \); \( \alpha \)=contact angle of the pipe with the wellbore........................................................................................15
Figure 2.9: Contact area of CwD (right) is greater than contact area of drill pipe (left). \( A_1 < A_2 \); A=contact area of the pipe with the wellbore ..............................................................................................15
Figure 2.10: Penetration depth onto filter cake, penetration of CwD (right) is less than penetration of drill pipe (left). \( d_1 > d_2 \); d=penetration depth into the filter cake..............................................................................................................................15
Figure 2.11: Wells drilled with CwD outperformed conventional wells in gas production ................................16
Figure 2.12: Three basic steps of plastering effect generation.............................................................................18
Figure 2.13: Additional contribution of plastering effect to borehole strength. LOT result for CwD (right) is more favorable than LOT result for conventional drilling (left) .................................................19
Figure 2.14: Comparison of formation damage with CwD and conventional drilling .....................................20
Figure 2.15: Wear band installed under casing coupling ......................................................................................22
Figure 2.16: Rigid centralizers...............................................................................................................................23
Figure 2.17: Shear stress-Shear rate curve for different fluid types.......................................................................26
Figure 2.18: Representation of annular space with a slot approximation (a) concentric annulus, (b) eccentric annulus....................................................................................................................................................28
Figure 3.1: The framework showing order of functions in Pre-Processor, Solver and Post-Processor of CFD..............................................................................................................................................34
Figure 4.15: 3-d velocity contour plots for 30% eccentric annulus and with inner pipe rotation @ 80 rpm....84
Figure 4.16: 3-d pressure contour plots for 60% eccentric annulus and with inner pipe rotation @ 80 rpm. ...85
Figure 4.17: 3-d velocity contour plots for 60% eccentric annulus and with inner pipe rotation @ 80 rpm....86
Figure 4.18: Velocity distributions across annulus for rotating inner pipe cases. ..................................................87
Figure 4.19: Pressure drop in the annulus profiles for rotating inner pipe cases. Pressure drop near wellbore. ..................................................................................................................88
Figure 4.20: Pressure drop in the annulus profiles for rotating inner pipe cases. Pressure drop near casing. ...89
Figure 4.21: Comparison of pressure drop in the annulus profiles for stationary inner pipe and rotating inner pipe cases. .................................................................................................................90
Figure A.1: 3-d pressure contour plots for 10% eccentric annulus and without inner pipe rotation.. ............100
Figure A.2: 3-d velocity contour plots for 10% eccentric annulus and without inner pipe rotation. ..............101
Figure A.3: 3-d pressure contour plots for 20% eccentric annulus and without inner pipe rotation. ..........102
Figure A.4: 3-d velocity contour plots for 20% eccentric annulus and without inner pipe rotation. ..........103
Figure A.5: 3-d pressure contour plots for 40% eccentric annulus and without inner pipe rotation. ...........104
Figure A.6: 3-d velocity contour plots for 40% eccentric annulus and without inner pipe rotation. ..........105
Figure A.7: 3-d pressure contour plots for 50% eccentric annulus and without inner pipe rotation. ..........106
Figure A.8: 3-d velocity contour plots for 50% eccentric annulus and without inner pipe rotation. ..........107
Figure A.9: 3-d pressure contour plots for 80% eccentric annulus and without inner pipe rotation. ..........108
Figure A.10: 3-d velocity contour plots for 80% eccentric annulus and without inner pipe rotation. .........109
Figure A.11: 3-d pressure contour plots for 10% eccentric annulus and with inner pipe rotation @ 80 rpm..110
Figure A.12: 3-d velocity contour plots for 10% eccentric annulus and with inner pipe rotation @ 80 rpm. .111
Figure A.13: 3-d pressure contour plots for 20% eccentric annulus and with inner pipe rotation @ 80 rpm. .111
Figure A.14: 3-d velocity contour plots for 20% eccentric annulus and with inner pipe rotation @ 80 rpm. .112
Figure A.15: 3-d pressure contour plots for 40% eccentric annulus and with inner pipe rotation @ 80 rpm..114
Figure A.16: 3-d velocity contour plots for 40% eccentric annulus and with inner pipe rotation @ 80 rpm. .115
Figure A.17: 3-d pressure contour plots for 50% eccentric annulus and with inner pipe rotation @ 80 rpm..116
Figure A.18: 3-d velocity contour plots for 50% eccentric annulus and with inner pipe rotation @ 80 rpm. .117
Figure A.19: 3-d pressure contour plots for 80% eccentric annulus and with inner pipe rotation @ 80 rpm..118
Figure A.20: 3-d velocity contour plots for 80% eccentric annulus and with inner pipe rotation @ 80 rpm..119
Figure A.21: Velocity distributions across annulus for all non-rotating inner pipe cases. ..........................120
Figure A.22: Pressure drop in the annulus profiles for all non-rotating inner pipe cases............................121
Figure A.23: Velocity distributions across annulus for all rotating inner pipe cases................................122
Figure A.24: Pressure drop in the annulus profiles for all rotating inner pipe cases .................................123
# LIST OF TABLES

Table 3.1: Set of data for input parameters. .................................................................................................................. 45
Table 3.2: Finalized input parameters for concentric and 0.5 eccentric case. .................................................................. 49
Table 3.3: Description of the mesh configurations used in grid independence study. ........................................................ 50
Table 3.4: Cases simulated in this dissertation. ............................................................................................................. 54
Table 3.5: Frictional pressure drop received in CFD, in numerical and analytical solutions. .............................................. 59
ACKNOWLEDGEMENT

This thesis became possible with suggestions and encouragements of several precious people during my two years of master’s study. Without their support, this study would not have been possible.

First and foremost, I would like to express my deepest and sincerest thanks to my advisor, Dr. Alfred (Bill) Eustes for sharing his wisdom and supervising me during my study. I am very grateful for this experience and guidance he provided. I would also like to thank to Dr. William Fleckenstein and Dr. Azra Tutuncu for their valuable input during my degree and for the motivation they provided. I am also sincerely grateful to Dr. Francisco Sanchez for guiding me with his deep knowledge on this topic and providing prompt and objective answers to my questions.

I would like to thank my colleagues Yasin Demiralp and Moji Karimi for the technical assistance they provided in every step of this project.

I am also very grateful to my colleagues in Golden, especially Hakan Corapcioglu, for never letting me down through my time spent here. The motivation they provided helped me immensely since the very first day I arrived in Golden.

I also would like to express my appreciations to Turkish Petroleum Corporation for providing me a full scholarship and financial support during my degree education.

Lastly, my deepest appreciations go to my family. Without their unlimited love, support and confidence in me throughout my life, I could never make it to where I am today.
CHAPTER 1

INTRODUCTION

Globally rising demand for oil and natural gas, and an increasing rate of depletion in producing reserves, led the oil and gas industry to utilize resources residing in more challenging environments. These environments - including deep-water environments, depleted zones and high pressure high temperature (HPHT) zones - required advancements on current drilling technologies to extract oil and gas.

Casing while Drilling (CwD) technology stands as a response to practical needs of the industry. The CwD method is operated by drilling the well with a specialized drilling bit attached to the casing string instead of drilling the well with conventional drill string. The innovative CwD method eliminates the need for wiper trips prior to casing/cementing operations, because the casing string is already run in the hole as the well is being drilled. Therefore, it helps to reduce nonproductive time in the drilling operations. There are two different types of CwD application. They are retrievable CwD system and non-retrievable CwD system.

The CwD technology with retrievable system was available since 1999 (Warren et al., 2004). Conoco-Philips was the first to imply the retrievable CwD technology in Lobo Trend in South Texas. On the other side, Mojarro et al., (2000) state that Pemex pioneered the technology in June 1996 with drilling with casing/tubing in Burgos Basin, which is a continuation of Lobo field in the south Texas. Shell was the other company to apply the non-retrievable CwD technology successively in the same basin as a part of underbalanced drilling with casing operations (Gordon et al., 2005). Over the last decade, CwD became more common. The successive results of the massive CwD projects in Lobo Trend in south Texas have boomed the popularity of the technology and more research was conducted. Following that, it has been applied in offshore drilling projects and in horizontal drilling with specialized steerable motor assemblies. As the technology became widespread, unique features have arisen. Two of these features are “the conjectured plastering effect” and equivalent circulating density (ECD). Plastering effect and higher
ECD as compared to conventional drilling, two inherent features of CwD, have been the focus point in the recent studies.

As a result of the casing being forced against the wellbore as it advances, the plastering effect is generated in the form of the drilled solids and bridging materials plastered against the borehole and packed into the filter cake with the constant motion of the casing string. The plastering effect is conjectured to provide a better filter cake quality and improves the borehole strength, which enables drilling through the highly porous zones with tendency to well instability and loss circulation issues. Presently, the plastering effect can be defined as the qualitative contribution to wellbore stability and increment in wellbore strength (hoop stress around wellbore). On the other hand, higher ECD of the CwD is mainly led by the narrow annular space in the wellbore. Experiences and studies dictate that for the best result, detailed study on these two features must be conducted. Along with several supplementary factors including formation characteristic, particle size distribution, in-situ stress distribution; the optimum combination of borehole geometry, drilling dynamics and flow regime is the key contributor to maximize the success of the Casing while Drilling technology.

Based on field experience, the plastering effect of CwD technology has been claimed to be advantageous. According to Watts et al., (2010) Casing while Drilling (CwD) technology stands as an engineered approach to significantly improve wellbore strength due to plastering effect. In that study, the plastering effect of CwD is addressed as the solution to lost circulation and wellbore failures, especially in depleted zones. Although many parameters governing plastering effect have been addressed, limited research has been done on their magnitude and the way they impact the final result. A finite element modeling study by Arlanoglu (2011) attempts to illustrate the relationship between the advantages of this technology and smearing effect by investigating the hoop stresses at fracture sealing with certain assumptions for the accounts of cutting size and transportation, crack sizes and in-situ stress distribution. With being empirically proven, Watts et al. (2010) and Karimi et al. (2011) have proposed that the plastering effect of CwD utilizes high annular velocity, pipe rotation and diameter ratio so that the drill
cuttings are smeared against the wellbore to form a stronger and effectively sealing filter cake. While several benefits are listed as above, high annular pressure losses, resultantly, higher equivalent circulating density (ECD) for CwD operations, has been addressed as a downside of the technology considering the tendency to easily damage the formation. Analogous approach to pressure loss analysis in slim-hole wells studies verified the impression that high ECD as a natural part of CwD is a definite disadvantage. However, recent researches focused on the additional pressure exerted on the wellbore as an aid to build impermeable filter cake isolating reservoir from borehole. The link between annular pressure and velocity profile, and plastering effect is still being sought. Defining these unknowns will clarify the extents and remedial applications of this technology.

Analysis of fluid flow in the annular space is not a new topic in the oil industry. The initial work on this topic included experimental studies and analytical models. The first work covered the investigations of flow field of laminar flow of Newtonian fluid in concentric annulus. The evaluation continued with introducing non-newtonian fluids, uniform eccentricity, and the eccentricity varying with depth and turbulent flow regime. All these studies were limited to mathematical models and experimental studies. The studies were mostly conducted through investigations on frictional pressure losses, velocity profile, viscosity distribution, shear rate and Reynolds number. The developments in computational sciences and the need for flexibility in capturing effects in various scenarios with less effort fostered the implementation of Computational Fluid Dynamics (CFD) in fluid flow field interpretation. Particularly in the oil industry, it is common to apply this method to analyze fluid flow in annuli.

1.1 Motivation of Study

The novelty of this study comes from simulating the specific CwD geometries and flow conditions, and discussing their results to set a background for further studies solely linking these results to the plastering effect of CwD. The models created through this study intend to investigate annular pressure losses, velocity profile and their trends in annuli with a high diameter ratio (CwD conditions)
and with some assumptions. The main focus is on the velocity and pressure responses as a function of eccentricity and presence of continuous rotational motion of inner pipe.

In this thesis, annular pressure and flow velocity profiles for the specific vertical CwD well geometry with a given eccentricity and narrow annular clearance (given borehole diameter to pipe diameter) are investigated. This study is a modeling project conducted through computational fluid dynamics. The finite volume models are created through ANSYS-Fluent, and they are used to illustrate annular pressure and fluid velocity profiles of drilling fluid as a function of eccentricity in specific CwD well environment. **Figure 1.1** illustrates velocity profiles in casing and in an eccentric annulus.

### 1.2 Objectives

The specific objectives of this dissertation are as follows:

- Study the effect of eccentricity and presence of continuous uniform motion of inner pipe on mud displacement processes from the point view of frictional pressure loss and fluid flow velocity in CwD annuli by simulating the case through cutting edge technology Computational Fluid Dynamics (CFD).
- Clarify some unknowns for the CwD annular hydraulics and contribute to understanding of flow field occurring during CwD operations.
- Set a background stage for a potential further study on establishing the link between annular pressure distribution and conjectured plastering effect, with associated wellbore strengthening.

### 1.3 Thesis Outline

In this chapter, introductory information about the CwD technology, the plastering effect and drilling fluid hydraulics associated with it are presented. Also the motivation and the objectives of the study are available. In this chapter, the uniqueness and novelty of the study are mentioned.
Chapter 2 provides a literature review on the CwD application. Presented are an overview of the technology, current status in practice, processes, advantages and limitations. Also, the plastering effect is described with factors affecting it and advantages associated to it. Chapter 2 covers a broad review of hydraulics for CwD as well. Theoretical information about fluid rheology, annular flow of power law fluids, specific CwD conditions and the pressure drop in annulus are briefly discussed.

Chapter 3 gives an introduction to the CFD model. This chapter covers the geometry and grid generation, assumptions and scenarios about the model, simulation procedures, governing equations, boundary conditions and input parameters.

Chapter 4 presents and interprets the results from the simulation runs. The graphs and data are discussed in details. In this chapter, verification of results by numerical solutions is available.

Chapter 5 presents the conclusion and a discussion of the work done in this thesis, as well as recommendations for future studies are presented.

Figure 1.1: Eccentric flow in annulus (Chin, 2011).
CHAPTER 2
LITERATURE REVIEW

In this chapter, aspects related to the Casing while Drilling technology are introduced with the benefits and limitations explained. Furthermore, features of the plastering effect are presented. Lastly, findings from a literature research on fluid rheology and chronological progress of annular fluid flow investigation are illustrated.

2.1 Casing while Drilling Technology

Casing while Drilling (CwD) technology is an innovative drilling technique that eliminates the need for the conventional drill string consisting of drill pipes, heavy weight drill pipes and drill collars in the drilling operations. Instead, this method utilizes a special bottom hole assembly connected to casing (Sanchez and Al-Harthy, 2011). Figure 2.1 shows the differences in the drill string geometry. Over the last decade, the CwD technology has worked satisfactorily in the contemporary drilling environments and has gained great interest, as it decreased non-productive time (trips, casing operations, etc.) of drilling operations. Along with the implementation of this technology, numerous facts about drilling with a larger sized diameter tubular have appeared. Several authors in their studies suggest that benefits associated to the CwD technology can be listed as; improved well economics, borehole stability, wellbore integrity, reduction in number of casing/liner strings, personal safety and overall drilling efficiency (Sanchez and Al-Harthy, 2011; Karimi et al., 2011; Watts et al., 2010; Karimi et al., 2011; Rosenberg et al., 2010; Karimi et al., 2012). These benefits accelerated the research and development initiatives on the technology. Most of these benefits are supposedly related to the plastering effect of CwD.

Plastering is the action of cuttings and solids in the drilling fluid being smeared and compressed against the borehole wall creating a semi-impermeable barrier. The plastering effect of the CwD is the physical contribution to the wellbore strength (increment in hoop stresses around the wellbore) and wellbore stability. This effect performs under certain conditions, including rotational motion of large
sized tubular and highly pressurized annular system. High pressure in the annulus is responsible for creating microfractures in the borehole, especially on the existing filter cake. Additionally, the smooth and continuous rotational movement of the casing string as it rotates against the wellbore tends to pack cuttings and drilling fluid additives into the readily opened small cracks and form a thicker and stronger filter cake. This process helps the CwD technology to minimize problems in highly porous troublesome zones. Reduction in number of casing strings, higher – quality wellbores in gauge, benefits with well control and wellbore stability are empirically linked the plastering effect.

2.1.1 Casing while Drilling Process

CwD essentially drills the hole by using a casing string as the drill string. Since actual drilling is conducted with the casing itself, the well is automatically cased and ready to cement once the target casing depth is reached. Usually regular casings in API standards are durable enough to satisfy operating conditions. Based on the weight on bit (WOB) requirements and torque-drag conditions, special accessories such as wear bands and torque rings can be installed in connections (Gupta, 2006).

![Figure 2.1: (a) Non-retrievable CwD assembly, and (b) Conventional drilling assembly (Karimi et al. 2012).](image-url)
CwD technology must be employed with several modifications in the rig set up. Most importantly, an automated drive system must be mounted to the top drive mechanism to safely connect individual casings to the string. This system is responsible for pipe handling, connection, transfer of motion and transfer of fluid flow. The surface casing drive system mounted to top drive grabs the individual casing internally via a spear ball and externally via a slip mechanism, and makes connection and transfer of fluid through inside casing. Also it seals the casing and prevents leaks in drilling fluid transfer (Warren et al., 2004). Figure 2.2 shows a casing drive system. In addition to the casing drive mechanism, size and capacity of the rig can be reduced. Since trips will be eliminated, the hoisting system elements can be modified. Mud pump capacities can be decreased as well. Overall, these changes make the rig more practical by making it easier and faster to transport and to set up. The wellhead equipment and blowout preventer configuration must also be appropriate for large sized tubular (Gupta, 2006).

The CwD technology is commonly practiced with two methods; non-retrievable CwD and retrievable CwD. Kenga et al. (2009) described the operational procedure for the non-retrievable CwD system, which includes a drillable drilling bit / casing shoe attached to the casing string with float collar rigid stabilizers installed on it. Once the target depth is reached, cutters and steel blades are pushed out of the drilling path and the aluminum portion of the bit stays in place. Cementing is conducted through this portion, and the new section starts by drilling through the remaining parts of PDC bit in place and rat hole with the new drill string. The simple drill string geometry of the non-retrievable system is shown in Figure 2.3 with a string connected to a drillable PDC bit / casing shoe.

On the other hand, the retrievable CwD system utilizes a custom bottom hole assembly (BHA) set up assembled to casing string with drill lock assembly. The specific BHA includes a pilot PDC bit and underreamer. Optionally it can include a downhole motor, MWD tools and a configuration of stabilizers. The PDC bit drills the pilot hole and underreamer enlarges the wellbore to its final shape. The drill lock assembly transfers motion from casing string to BHA. The retrievable CwD offers the flexibility of changing drill bit and directional drilling (with directional drilling tools add-on to the BHA). Based on the
necessity, BHA can be pulled and run into the hole by wireline or drillpipes. **Figure 2.4** shows a model BHA for the retrievable CwD system. The selection of the proper method is based on drillability of the interval of interest with single-run (can be estimated according to the previous bit records, drilling parameters and logs) or trajectory requirements of the wellbore (Kenga et al., 2009).

![Casing Drive System mounted to top drive](image1)

**Figure 2.2**: Casing Drive System mounted to top drive (Warren et al., 2004).

![Drillable PDC bit](image2)

**Figure 2.3**: Drillable PDC bit (Kenga et al., 2009).
From an engineering standpoint, the application of this technology requires deep understanding of the technology and a systematic approach in drilling operations. In a field case study by Sanchez et al, (2011), the authors suggested applying a strict procedure in the operation. They suggested dividing the operation into three phases: pre-operation phase, drilling phase and post-operation phase. In the pre-operation phase, the limitations and risks induced by field properties must be well examined. Approximate drilling fluid parameters and wellbore geometry are designed in this phase. Also, correlation to the other wells helps to detect problematic sections. During the actual drilling phase, the operation must be tracked meticulously. Using fundamental drilling engineering concepts and effective practices, real-time parameters must be managed and key components must be updated. Surface pressure, drilling fluid returns, ROP, bit performance, and torque and drag are important parameters to control in this step. The post-operation phase must aim to perform efficiency review sessions and to enhance the applied technology with the new ideas and the contribution of various technologies based on evaluation of the
operation. The authors claimed that successful results and development of this technology are not guaranteed if these conditions are ignored (Sanchez et al., 2011).

2.1.2 Advantages of Casing while Drilling Technology

CwD technology has been a helpful method in many successful field applications. There are only a few theoretical studies to back up the benefits of this application; however, the achievements of this technology are shown in published case studies. These achievements can be listed as; success in drilling through lost circulation zones and active shale zones, enhanced well economics, wellbore strengthening, and lost circulation prevention. Most of these advantages are in some way linked to each other and by means of them; the CwD method stands as a favorable approach over the conventional drilling methods (Karimi et al., 2011). The following benefits of CwD will be discussed in this section:

- Enhanced well economics and well delivery time
- Elimination of swab and surge effects
- Rig adaptation and HSE
- Wellbores in gauge
- Improvements on production
- Wellbore strengthening
- Lost circulation reduction

2.1.2.1 Enhanced well economics and timing

CwD technology enhances well economics and well delivery time, especially in the problematic wells with high tendency of lost circulation and unstable wellbore. As this approach is implemented, the unproductive time spent on casing running and cementing is strikingly reduced. Since the casing is already in place, the necessity of wiper trips and well conditioning for casing running operation is eliminated. Figure 2.5 represents a graph from a successful implementation of CwD in Nigeria. The
figure shows how the drilling curve is developed with the introduction of the CwD method in Akamba-2 well. In addition to the necessity of wiper trips and well conditioning, most of the well control events, loss circulation and related unpredicted non-productive time events are avoided. On the counter act, the specific modifications on drilling rig, special bits, unique casing connection wear bands, materials and accessories increased the operational cost once the cost breakdown of the drilling project was investigated. Nonetheless, an overlook of the entire project verifies the merits of this method with reduced operational time and minimized risk of wellbore related incidents (Sanchez and Al Harthy, 2011). All these aspects favor the well economics. In a field based study by Sanchez et al. (2011) questioning the drilling efficiency optimization of CwD, it was shown that the proper implementation of the technology can accelerate the well delivery time significantly, reaching up to 58% and reduce the well cost more than 25% in cost per meter.

Figure 2.5: Improved drilling curve of Akamba -2 well. Solid line represents actual operation and dashed line represents planned operation (Kenga et al., 2009).
2.1.2.2 Elimination of swab and surge pressure effects

Eliminating the drill-pipe trips serves for the wellbore safety as well. The non-retrievable CwD approach offers a trip-free drilling operation; whereas, in the retrievable CwD system, only the BHA is tripped through casing. From the drilling engineer’s standpoint, this will positively affect the range of operable mud weight window by removing the swab and surge safety margins, and providing flexibility with mud weight. In the operator’s perspective, this advantage will help to push limits on casing setting depth selection curves. (Karimi et al., 2012) It can even help to reduce the number of casing strings and finalize the well with a larger production casing, which may assist the production engineer with greater sized production tubes. Figure 2.6 shows how the elimination of safety margins can work for casing design as the number of casing strings is reduced.

Figure 2.6: Effectively modified operational mud weight window after removing trip margins (Karimi et al., 2012).
2.1.2.3 Rig adaptations and HSE

A custom designed rig for the CwD applications is more practical and more efficient than the conventional rig set up. The rig adaptation guidelines for CwD eliminate the need for great horsepower in rig units. Hoisting systems and mud pumps can be redesigned considering the specific conditions of CwD. Also, these types of rigs require capability to lift only a single joint, reducing the mast height. After all these modifications, the rig turns into cutting edge technology machines, which are more practical, and easier to move and rig up. In addition to that, the time spent on mobilization, transfer and rig up, and logistic services are improved. Besides ease in operability, the well site safety is improved with automated systems and incidents while handling pipes are lessened (Gupta, 2006).

2.1.2.4 Wellbores in gauge

In drilling operations, the gauged holes are preferred because they enable effective cementing operations and improved wellbore cleaning with superior hydraulics. The CwD pipe geometry tends to create a gauged well by means of the smooth rotational motion of casing. Figure 2.7 represents an example of the difference between a conventionally drilled well and a casing drilled well. The physical explanation beyond CwD-offered better wellbores consists of casing contact angle and area of the casing in contact with wellbore, and penetration depth into filter cake. During the CwD processes, the casing string hits the borehole with a smaller contact angle and greater contact area. This action combines the side force and momentum of the pipe with grinding effect to generate a more circular wellbore; and potentially help to fill in washouts and breakouts. From the penetration depth into filter cake standpoint, when compared to drilling with drill pipe, casing will have the same force due to rotation of the pipe; yet, the area on which it is applied is greater. Thus, pressure applied on the wellbore by physical contact of casing will be moderate and that will rub filter cake instead of damaging through it. In their study, Karimi et al. (2011) analyzed the casing and wellbore geometry in detail. Figure 2.8, 2.9 and 2.10 compare CwD and conventional drilling based on contact area, contact angle and penetration depth into filter cake.
2.1.2.5 Improvements on production

One of the striking benefits of this technology is observed in reduced formation damage and better production performance. Minimizing the time that drilling fluid is in contact with formation is the reason behind this benefit. Good wellbore quality with improved seal and isolation of formation dictates
smaller skin effect. Tessari et al. (2006) illustrated one of the most obvious examples of this claim in their study. According to the authors, the CwD method was employed in a south Texas gas field to develop a field producing from depleted zones. The wells drilled with CwD successfully performed in the first year of production. The authors relate this fact to the plastering effect of CwD and its inherent properties. Consequent to the plastering effect, good isolation between wellbore and formation was provided and cement operation was conducted quickly and effectively with minimized lost circulation to formation. Figure 2.11 shows the success of wells drilled with CwD in yearly production point.

![Figure 2.11](image)

Figure 2.11: Wells drilled with CwD outperformed conventional wells in gas production (Tessari et al., 2006).

### 2.1.2.6 Wellbore strengthening and lost circulation reduction

The most popular advantages of CwD are improvement in borehole stability, facilitations in eliminating lost circulation and drilling through active shale zones. Many case studies from different fields all across the world pointed out the remedial effects and point out the success of this technology (Sanchez et al., 2010; Lopez et al., 2010; Dawson et al., 2010; Gallardo et al., 2010; Beaumont et al., 2010; Torsvoll et al., 2010; Rosenborg et al., 2010; Costeno et al., 2012). A striking application was reported by Sanchez et al. (2010) in Fiqa formation in Oman. This study was conducted based on field results and it questioned the drilling efficiency optimization of CwD. It is concluded that CwD is the best
way to mitigate problems associated to swelling active shale zones such as stuck pipe, back reaming necessity and unexpected casing setting depth decisions. Their field results supported this opinion similar to the other reported results.

In numerous occasions, a majority of the benefits are linked to the plastering effect of CwD. This is also expressed in several field studies consecutively. Yet, only a few researchers have focused on the physics of this process. The resources yield only empirical data about generation processes of the plastering effect. The data available suggests qualitative results rather than quantitative information. There are no quantitative results from laboratory experiments, mathematical modeling, simulation or experimental efforts in the field. Therefore, factors controlling the process and advantages associated with the plastering effect have not been defined in clear lines. The study by Fontenot et al. (2004) is one of the pioneer works, which expressed this impressive effect. However, Watts et al. (2010) hold the most relevant study on the plastering effect of CwD. This study is based on field results obtained in Piceance Basin and Alaska. According to the study, the remedial treatment of CwD in lost circulation reduction and the wellbore strength improvement were only dependent on plastering effect. The study failed to present a detailed explanation for the mechanism of these processes. Instead it discussed wellbore strengthening responses of different bridging materials tested in the drilling fluid system. The consequences of the tests led the authors define the plastering effect as a factor raising the effective fracture gradient by smearing and plastering cuttings into the wellbore and/or increasing the hoop stress around the bore hole as a crack is formed and particles are forced into the gap.

Furthermore, in their study, Karimi et al. (2011) claimed that the smooth rotational movement of casing in combination with high annular pressure reinforces the impermeable interface by pushing the particles in the drilling fluid into it. This impermeable interface is responsible for benefits such as gauged wells, improvements on production, wellbore stability, wellbore strengthening, and lost circulation reduction. The improved interface in this study was linked to the plastering effect. Figure 2.12 schematically summarizes the plastering effect in three phases. In the first step (left figure), casing
contacts the bore wall with force as it is runs into the borehole. In the second step (middle figure), filter cake forms up with drilling fluid mud is smeared into the formation. In the third step (right figure), the filter cake and cuttings are plastered against the borehole wall, sealing porous formation.

Figure 2.12: Three basic steps of plastering effect generation (Karimi et al., 2011).

In parallel to Watts et al.’s definition, Arlanoglu (2011) supported the opinion that the plastering effect is analogous to the stress caging idea. In his dissertation, the author modeled the creation of micro fractures, accumulation of the bridging materials in these fracture mouths, and then the increase in hoop stresses around the wellbore (Arlanoglu, 2011). Yet, some points about this idea must be clarified. The role of high equivalent circulating density to create new fractures is physically questionable. The fluid flow would prefer the easier path, which is the axial flow instead of flowing into the formation. In addition to that, bridging materials, pore size and shape, particle size distribution, rotational motion of pipe, formation characteristics and fluid hydraulics must cooperatively favor the conditions that support the generation of plastering effect. These listed conditions must be well-ordered so that ideal packing theory successfully applies. A study by Vickers et al. (2006) defines the ideal packing theory (IPT) as a method of improving the bridging efficiency for drilling fluids. The authors pointed out the relationship between the particle size distribution (PSD) and the pore size distribution. According to this theory, ideal packing occurs when the percent of cumulative volume versus the square root of the particle diameters forms a straight line. This approach assigns excessive importance to drilling fluid additive properties and formation properties. As drilling fluid type, the water based mud with shear thinning effect favors the
packing action, regarding its effective mud cake generation and rheological properties. The effect of drilling fluid must be coupled with the right PSD and the concentration of solids. In such an operation, the solid concentration of the drilling fluid is regulated artificially during drilling fluid preparation. Apart from that, particle concentration and shapes are also dependent on the bit type and lithology properties. Watts et al. (2010) showed that in Piceance basin and Alaska examples, loss control materials ranging from 100 microns to 2000 microns in size have performed successfully. **Figure 2.13** illustrates the result of a successful application from this study. Regarding formation characteristics, the porous structure of formation favors the process. It is important to note that CwD with the plastering effect is not an operational cure to the extensive fractures prone to total fluid loss to the formation. The plastering effect usually appears after filling the micro fractures, which were either already in place or created by the extensive pressure applied in annulus towards the wellbore.

![Figure 2.13](image_url)

**Figure 2.13:** Additional contribution of plastering effect to borehole strength. LOT result for CwD (right) is more favorable than LOT result for conventional drilling (left) (Watts et al., 2010).

Rather than the stress caging theory, a more commonly accepted explanation for this action is the contribution of the side force and the momentum. According to this approach, the most essential component is the physical contact between the casing and the wellbore. The flow type, well geometry (eccentricity, drill string geometry) and pipe stiffness are contributors to generate sealing filter cake. Theoretically, a successful result is possible with the larger diameter casing (smaller contact angle, larger
contact area so that the mud cake will not be damaged by movement of string) and a greater ratio of casing outer diameter to hole size. In the study by Karimi et al. (2011), the ratio was suggested to be between 0.75 and 0.90 for the best results. However, diameter ratio selection is a case oriented process. For this reason, in different wells with distinct formation types, formation fluid properties, hydraulics and mud design; the ratio is expected to be different regarding operational troubles. As the pacesetter trend, American Petroleum Institute (API) recommended casing selection charts offer a realistic approach to the research studies (API Specification 5CT, 1999).

Overall, the plastering effect aided impermeable seals to mitigation of drilling-induced formation damage, improved wellbore stability, and reduced lost circulation (Karimi et al., 2011). A schematically represented well model with formation damage demonstration for both cases is available in Figure 2.14.

![Figure 2.14: Comparison of formation damage with CwD and conventional drilling (Karimi et al., 2001).](image)

After the plastering is induced by stress caging, physical contact or combination of both, the most applicable method to check the presence of the plastering effect is formation integrity or leak off tests. As in Figure 2.13, these tests will indicate the advancement in the wellbore strength and generation of a strong seal. Based on the results, the casing design (setting depths) can be modified by utilizing the increased wellbore strength. Another method employed by Fontenot et al., (2004) is testing the physical properties of sidewall cores taken from the wells drilled with casing. The authors verified the presence of plastering using this method.
A final remark about the plastering effect process can be drawn about the casing rotational motion. The process is absolutely dependent on the rotational motion of casing string. Therefore, as the strong filter cake generation is investigated, the most effective plastering is expected to happen where the string is in compression and contacting the borehole. This point is a function of the axially varying pipe eccentricity that is monitored by string stabilization, pipe stiffness and diameter ratios. As discussed above, the lithology and the drilling fluid with appropriate pressure, velocity and rheological properties (cutting transport capacity, gel strength, mud cake generation property) are completing factors to the physical effect. Since the whole process depends on the rotation of the large sized tubular, retrievable system with downhole motors introducing sliding mode of directional, non-rotating string or minimal rotation of string will not promote the process; instead, a non-retrievable system with higher rotational speed is preferred.

2.1.3 Limitations of Casing while Drilling Technology

Although this technology is populated with several advantages, it has major restrictions as well. Only few studies objectively analyzed these issues. Some of these limitations are eased with practical solutions and many of them are in the process of being solved through temporary solutions or alternative methods, but the technology needs more development in these areas.

In the operational standpoint, weight on bit applied and pipe rotational speed are restricted. Regular oil field casings are manufactured for the static conditions in wellbore. As they are exposed to the dynamic conditions with the rotational motion; cyclic fatigue, torsion cycles, compressive loads and torsional requirements must be redefined (Galloway, 2004). The physical properties of tubular and connections assign weight on bit capacity. Usually, the CwD string utilizes no special BHA to provide to WOB; therefore, the lower part of the casing string is in compression. In the application, limits of buckling are obtained through finite element models in the pre-execution phase. The models tend to test the stress capabilities of the casing. Alternatively, heavy casings with thick walls can be used in the
critical sections. This will not only prevent failure due to buckling but also reduce the significance of distributed wear. Tungsten carbide hard faced wear band installation below connections is another option to preserve casing’s original physical properties. Figure 2.15 shows installed wear band in casing connection. Lastly, the combination of stress levels in the pipe and number of cycles determine the reduction in fatigue tolerance. Fatigue testing and drilling performance usually suggest that the buttress type connection performs sufficient fatigue tolerance (Warren et al., 2004; Gupta 2006).

![Wear band installed under casing coupling](image)

Figure 2.15: Wear band installed under casing coupling (Warren et al. 2004).

Apart from the tubular physical durability concerns, cementing, particularly centralization of casing string is a major operational concern. The regular spring type centralizers fail to withstand the dynamic downhole conditions. Commonly, the rigid centralizers are used for wear management and centralization for cementing. Figure 2.16 shows a rigid centralizer that can be used in CwD string. While they are durable to downhole conditions, the implementation of rigid centralizers is limited by the additional torque and cost. Then again, rotating casing during displacement of cement may eliminate need for a centralizer. Additionally, the cementing tools in retrievable system have been known for a tendency to fail. In this system, once the BHA is pulled out, there is no tool in the string to prevent cement u-tubing. Waiting cement to set up in a pressurized system or displacement plug are other solutions to this issue.
Figure 2.16: Rigid centralizers (Warren et al., 2004).

The most severe limitation comes with the drilling fluid specifications and hydraulics behaviors. A common outcome of this disadvantage is differential sticking. The differential sticking is an issue especially in low pressured permeable zones. The rheological functions of drilling fluid must be designed to withstand this. More importantly, allocating the precise flow rate is on a very critical line. Equivalent circulating density (ECD) management is surely sensitive. The CwD application utilizes the characteristic narrow annulus to reach high annular fluid velocity and transport cuttings to surface wiping the annulus effectively. The narrow annular clearance puts the annular pressure loss in a vital position. Moderately low flow rates, in comparison to the conventional drilling conditions, satisfy the wellbore cleaning and ECD requirements; however, jetting action, cleaning bit face and cuttings from the bottomhole should be maintained as well (Gupta, 2006). The excessive flow rate leads to fractures in the formation. Although, some authors (Fontenot et al., Watts et al., Karimi et al., and Arlanoglu) mention benefits of having the high pressure profile in annulus on the generation of plastered, strong and high quality seal, it is still uncertain whether the high pressure is a risk or an advantage. Higher ECD is conjectured to initiate small fractures that are readily plugged by plastering effect combined with the stress cage mechanism. On the other hand, it is obvious that excessive ECD can ruin the uniformly shaped filter cake and create fractures. This area needs broad research verified with experiments, field results and modeling.
2.2 Review of Hydraulics

It is vital to understand the concept of annular frictional pressure drop and annular velocity due to slight clearance between the hole and the casing. To address these topics, drilling fluid hydraulics is captured in this section. Fluid models, drilling fluid properties and frictional pressure loss calculation studies are given along with current state of the research methods and the position of CFD in this evolutionary period.

2.2.1 Drilling Fluid Rheology

Wellbore hydraulics is a function of the rheology. Rheology, as a study, concerns the deformation and flow of matter. Fluids are subcategorized into the rheological models based on their response in the shear stress and shear rate curves. Shear stress is the equivalent force to maintain a particular type of flow. Shear rate is the ratio of the relative velocity of moving surface to adjacent surface over distance between them. The response of the fluid in a shear stress vs. shear rate curve indicates fluid type; Newtonian fluid and non-Newtonian fluid, rheological properties; viscosity, yield point and gel strength, and rheological fluid model; bingham plastic fluid, power law fluid and yield power law fluid.

Newtonian fluids have the simplest curve trend with a linear proportionality between shear stress and shear rate. Viscosity, the shear stress divided by shear rate, is constant over the complete range of shear rate. However, most of the drilling fluids are represented in the non-Newtonian fluids form. In the shear stress vs. shear rate curve for the non-Newtonian fluids, the trend does not behave linearly. Viscosity varies with shear rate and this leads to effective viscosity term, the viscosity defined at a specific shear rate. Distortion from the linearity in a shear rate vs. shear stress curve is a result of the time dependence and the shear rate dependence.

Typically, fluids of interest in the oil industry show sensitivity to shear rate rather than time. In order to describe these fluids better, several fluid models have been proposed. The bingham plastic model, power law model and yield power law model are commonly accepted. Figure 2.17 addresses bingham
plastic fluid, power law fluid and Newtonian fluid in a shear rate – shear stress curve. Shear stress and shear rate are calculated using data from Fann VG viscometer and the corresponding values are expressed in secondary axis.

Bingham plastic fluids yield a linear trend in the shear rate vs. shear stress graph. The slope of the line yields plastic viscosity, which is a function of the concentration, size and shape of solids and viscosity of the fluid phase. Separation from the Newtonian fluid is resulted by the stress required to initiate motion. In order to start the fluid moving, a level of stress must be applied and the stress required is called the yield point. Mathematically shear stress ($\tau$), plastic viscosity ($\mu_p$), yield point ($\tau_y$) and effective viscosity ($\mu_e$) are shown as given:

$$\tau = \tau_y + \mu_p \dot{\gamma}$$ \hspace{1cm} (2.1)

In this formula, the shear stress is greater than minimum shear stress. As the minimum shear stress is equal or greater than shear stress, then shear rate term is canceled since shear rate equals to 0.

Plastic viscosity and yield point are calculated using reading in Fann VG viscometer.

$$\mu_p(cp) = \theta_{600} - \theta_{300}$$ \hspace{1cm} (2.2)

$$\tau_y \left( \frac{lb}{100ft^2} \right) = \theta_{300} - \mu_p$$ \hspace{1cm} (2.3)

Finally effective viscosity of the fluid is given by

$$\mu_e = \mu_p + \frac{\tau_y}{\dot{\gamma}}$$ \hspace{1cm} (2.4)

Power law fluids show a parabolic trend in the shear stress vs. shear rate curve. Similar to the trend curve for Newtonian fluids, the curve starts from the origin and based on the value of power law index ($n$), it reflects a parabolic curve. As $n$ is greater than 1, power law fluid is considered as shear thickening and as $n$ is less than 1, it is shear thinning fluid. Bourgoyne et al. (1986) described power law index ($n$), consistency index ($K$) and apparent viscosity of a power law fluid as given:

$$n = 3.32 \log \left( \frac{\theta_{600}}{\theta_{300}} \right)$$ \hspace{1cm} (2.5)

$$K = \frac{511 \theta_{300}}{511^n}$$ \hspace{1cm} (2.6)

$$\mu = K (\dot{\gamma})^{n-1}$$ \hspace{1cm} (2.7)
Yield power law fluid model (Herschel-Bulkley model) is the closest to the typical drilling fluid. This model is a combination of bingham plastic fluid and power law fluid. Figure 2.17 graphically illustrates the resemblance. A yield stress is required to start flow similar to the behavior in bingham plastic fluids, and the trend of shear stress vs. shear rate curve is parabolic similar to the behavior in power law fluids. The yield power law fluids are mathematically more complex than bingham plastic fluids and power law fluids. Shear stress ($\tau$) and effective viscosity ($\mu$) of the yield power law fluid is given as:

\[
\tau = \tau_y + K\dot{\gamma}^n
\]

---

\[
\mu = \frac{\tau_y + K\dot{\gamma}^n}{\dot{\gamma}}
\]

---

Figure 2.17: Shear stress-Shear rate curve for different fluid types (Taken from Amoco Production Company – Drilling Fluids Manual).
2.2.2 Fluid Flow and Frictional Pressure Loss Analysis for a CwD annulus

The behavior of fluid flow field (pressure and velocity) is governed by the fluid rheology, wellbore geometry, and flow rate. In fluid flow applications with slim annular clearance, including CwD, pressure and velocity profiles alter significantly. The alteration occurs in support of high velocity in the annulus and high flowing bottomhole pressure. Fundamentally, the International Well Control Forum (IWCF) expresses the flowing bottomhole pressure as the summation of static bottomhole pressure and annular pressure loss (IWCF, 2006). As the terms in flowing bottomhole expressions are converted from pressure to equivalent mud weight, the equivalent circulating density (ECD) equation is formed.

$$ECD = \frac{\Delta P_{fric} + P_h}{D}$$ (2.10)

In this formula, $P_h$ stands for the hydrostatic mud pressure in the annulus, $P_{fric}$ stands for the sum of annulus pressure losses due to friction, and D stands for the depth. The annular pressure losses become significant, because ECD can occasionally be greater than formation fracture gradient. Normally, provided that the mud weight is kept constant, ECD is exposed to limited change, as the frictional pressure drop is not the superior in hydraulics design for conventional drilling geometries. However, the application of this approach to the CwD circumstances becomes perilous. In the CwD technology, frictional pressure losses predominate. The slimhole drilling applications show similarities to CwD regarding the tight annulus clearance and high annular pressure losses. One of the ways to control the dynamic bottom hole pressure (hence ECD) is through controlling the mud weight. As the mud weight decreases, hydrostatic pressure of the mud column in the annulus decreases. The rheological property of drilling fluid is another gadget to adjust in advance in order to lower the ECD. For instance, lowering plastic viscosity is an option to reduce the ECD. A final caution might be lowering the flow rate, depending on that, lowering velocities in the annulus and reducing the frictional pressure losses. Instead of individual application of any of these three methods; they must be optimized to obtain the best results. This is very crucial for the wellbore cleaning (Karimi et al., 2012).
Regarding this, assigning the proper flowrate considering the eccentricity and hole size to the pipe size ratio is of great significance. As a response to this significance, fluid flow in annuli has been studied analytically, numerically and with experimental models over recent decades. Lamb et al. (1932) were the first to mention the analytical approach on frictional pressure loss in a concentric annulus. Fredrickson et al. (1958) evolved the process by introducing the non-Newtonian fluids. Meanwhile, eccentric geometries were evaluated by two methods: the narrow slot approximation of annulus and the definition of eccentric annulus in bipolar coordinates. Both methods serve to transform eccentric plane to an easy to comprehend plane. Tao and Donovan (1955) were first to use a narrow slot approximated annulus approach in their analytical studies. This method was developed by other authors including Vaughn (1965); Mitsuishi and Aoyagi (1973); Iyoho (1981); Luo (1990). In 1989, Uner et al. (1988) developed Iyoho’s analytical solution to final form. Figure 2.18 illustrates the slot-approximated forms of a concentric and an eccentric annulus. As can be seen, in the concentric case, shear stress in x-direction is not included in the equation of motion. A one dimensional equation of motion satisfactorily represents the motion. However, in the eccentric annulus, the shear stresses are affected in both the x and y directions. This brings the concern of including both of these forces in the equation of motion. Haciislamoglu (1989), in his PhD dissertation reports that Tao and Donovan (1955) failed to include shear stress variation in x-direction in their slot approximated eccentric annulus analytical studies. Depending on that, the researchers following this study failed to have the correct equation of motion. On the other side, numerical studies, was first mentioned by Heyda (1959) utilizing the bipolar coordinate system to create eccentric annulus. Using different solution methods, the approach was employed by Redberger and Charles (1963) and Guckes (1975). Finally, Haciislamoglu and Langlinais (1990) utilized the finite difference method to numerically solve the problem.
In this thesis, Uner et al.’s (1989) equations from their analytical study and Haciislamoglu et al.’s (1990) equations from their numerical study have been used to compare and validate results obtained in simulations. They are further described in the model validation section.

Uner et al. defined the flow rate for a concentric case as

$$ q_c = \frac{\pi r_0^3}{2} \frac{n}{\gamma} \left\{ \frac{dP}{dy} \right\}^{\frac{1}{\gamma}} \left[ (1 + R_r)(1 - R_r) \right]^{\frac{1}{\gamma}} $$

(2.11)

The flow rate for an eccentric case is given as

$$ q_e = \frac{\pi r_0^3}{2} \frac{n}{\gamma} \left\{ \frac{dP}{dy} \right\}^{\frac{1}{\gamma}} \left[ \left( \frac{1 - n^2}{2\gamma - \pi R_r} \right) \right] F(f, n, R_r) $$

(2.12)

F, E, f and R, are the constants and they are the functions of eccentricity and radius.

Also, $q_e$ (m$^3$/s) is flowrate for eccentric case, $r_0$ (m) is radius of outer wall of annulus, K (Pa-s$^{-n}$) is consistency index, n is Power Law index, P (Pa) is pressure, and R, is radius ratio.
Haciislamoglu et al. equated the ratio of the pressure drop gradient in eccentric case to the pressure drop gradient in concentric case as given below. This equation is dimensionless and requires minimal constraints.

\[
R = 1 - 0.072 \frac{e}{n} (R_r)^{0.8454} - 1.5e^2 \sqrt{n}(R_r)^{0.1852} + 0.96e^3 \sqrt{n}(R_r)^{0.2527} \tag{2.13}
\]

where,

\[
R = \frac{\left(\frac{dP}{dy}\right)_e}{\left(\frac{dP}{dy}\right)_c} \tag{2.14}
\]

In this equation, \(\frac{dP}{dy}_e\) is pressure loss gradient for eccentric case, \(\frac{dP}{dy}_c\) is pressure loss gradient for concentric case, \(R\) is radius ratio, \(r_o\) (m) is radius of outer wall of annulus, \(r_i\) (m) is radius of inner wall of annulus, \(n\) is Power Law index and \(e\) is eccentricity.

Above, listed studies and the equations affiliated with them do not necessarily represent the drilling process of a well drilled with the CwD method. These studies can be regarded as a background step to understanding the physics beyond annular flow. Uniquely in CwD conditions, in addition to the eccentricity and flow regime, the inner pipe rotation plays a significant role. There exist a number of studies (both modeling and experimental) about simulating the hydraulics through annulus with inner-cylinder rotation. These studies mostly emphasize drill string rotation contribution to pressure losses. The most comprehensive study has been published by Escudier et al. (2001). This study introduces a numerical method for the non-Newtonian fluid flow in the eccentric annulus with inner pipe rotation. With the accurate implementation of the finite volume method, it is greatly acknowledged as one of the most descriptive studies in the field of contemporary flow analysis in realistic annulus. The authors redefined momentum and equations of motion in three dimensions including rotational movement. The shear rate and dependently viscosity profiles were introduced. Another study conducted by Diaz et al. (2004) reviewed ECD modeling in the CwD operations. Conveniently, these studies suggest that increasing rpm results in higher annular pressure losses. Ahmed et al., (2008) and Ahmet et al. (2010)
verified that the drill string rotation – annular pressure losses relationship is governed by fluid properties (rheology and density), flow regime, diameter ratio and eccentricity. These two experimental studies focused on the response of shear rate to pipe rotation. They further defined three effects counteracting each other to develop the frictional pressure loss in inner pipe rotating geometries. These three main factors are:

- **Inertial effect**: As a response to the rotation of extensively long drill string, the fluid elements with various velocities travel along streamlines through the annulus. Pipe wobbling and eccentricity fluctuations generate the inertial effects that contribute to pressure losses.

- **Secondary flows**: Higher rpm is likely to create centrifugal and shear instabilities. In that case, the secondary flows such as the Taylor vortices can be formed in the annulus and contribute to pressure losses.

- **Shear thinning effect of non-Newtonian fluids**: In contrary to inertial effect and secondary flows, the shear thinning effect, which decreases the effective viscosity as the shear rate increases, works against the increasing pressure losses because of the combination of axial and rotational flow through shear rate dependent apparent viscosity (Ahmed et al., 2008; Ahmed et al., 2010; Escudier et al., 2001).

Given these discussions, the fluid rheology – pipe rotation interaction turns into a determining parameter in a slim annulus. As the annular space is limited, it is probable to have high shear rates at moderate flow rates (Cartalos and Dupuis, 1993). The complexity of the problem; motion in axial, horizontal and tangential direction and the factors governing shear rate and dependently viscosity of the fluid, and developments in the computational technologies fostered the application of computational fluid dynamics (CFD) in this specific research area. Recently it was utilized several times to show effects of wellbore geometry, fluid rheology and similar factors on fluid flow (Ozbayoglu and Omurlu, 2006; Ogugbue, 2009; Sorgun et al., 2010; Mokhtari et al., 2012; Ernila, 2012; Osgouei 2012).
CHAPTER 3

METHODOLOGY

Non-Newtonian fluid flow in the eccentric annuli has been successfully captured by the Computational Fluid Dynamics (CFD) method. In this study, CFD is used on a model created through ANSYS – Fluent commercial software package with the purpose of illustrating annular pressure and annular velocity profiles of several specific CwD geometries.

3.1 Application of Computational Fluid Dynamics in Drilling Engineering

Drilling engineers and researchers have to deal with various types of fluid flow applications in numerous environments. One of the most common cases confronted is the flow of drilling fluid in the borehole. Practical solutions have been sought to this specific problem. As researchers investigate, various challenges for annular fluid flow modeling have been discovered. A major challenge is associated to the governing equations. The partial differential equations that are modeled, are highly nonlinear and difficult to solve. Classical approaches fail to solve the problems. Although the stationary wall bounded flow domain has a single equation used for the axial flow, aforementioned obstacles may exist even for this situation. Additionally having the rotating bodies, there is a high tendency to observe numerical instabilities led by the azimuthal flow coupling. Based on the problem specifications, robust algorithms and stable iterations are required. In addition to the governing equations, reflecting the complicated annular geometries to solutions is difficult to accomplish. A highly eccentric geometry, which may typically be encountered in an actual wellbore, must be satisfactorily reflected to the coordinate system such that the governing equations can be solved through it successfully. Finally, the yield stress influences and the rheology responses cause significant complications in obtaining solutions that match with the actual data. For the concentric rotating flows in vertical wells, the effect of rheology only appears through shear thinning; and that is responsible for observing reduced resistance. While the shear thinning
effect is present for the eccentric cases, certain nonlinear convective terms act on the governing equations, which alters the effective pressure gradient. These convective terms control the flow rate distribution in indirect ways as a result of shear thinning, secondary flows, and inertial effects. Difficulties cited above make the computational methods step forward. Computational fluid dynamics (CFD) approach stands as an essential and handy alternative with rigorous mathematics applied through it and state-of-the-art numerical analysis.

CFD is a widely applied technique integrating fluid mechanics, mathematics and computing science to simulate comprehensive science and engineering cases. The cutting edge CFD method has been used as a research tool, as an education tool and as a design tool in numerous high technology demanding fields including, aerospace, automotive engineering, biomedical science, civil engineering and sports. The feasibility of CFD results from the major advantages associated with it. CFD supports the experimental and the analytical approaches by providing an alternative cost effective means of simulating real fluid flow. Also, CFD is capable of modeling the flow conditions that are difficult to create in an experimental set up. With regards to result interpretation, CFD provides a better visualization with details and extensive information about fluid dynamics.

3.2 ANSYS – Fluent CFD Package

ANSYS Fluent is a graphical user interface created by ANSYS Inc. to facilitate its application and usage in handling very complex fluid flow problems for the first-time users. In this package, CFD analysis consists of three main elements:

- Pre-processor
- Solver
- Post-processor

These operators are conducted on a workbench that separates individual steps of an entire project. Figure 3.1 shows the division of these functions.
3.2.1 Pre-Process Stage

Pre-process stage hosts the creation of the problem. Specific conditions of the case are inputted in this stage. Namely, the steps constructed are:

- Model geometry,
- Flow areas and domains,
- Grid structure,
- Material properties,
- Initial and boundary conditions.
3.2.1.1 Creation of geometry

The first step in any CFD analysis is the definition and creation of geometry. Depending on the flow type (internal flow or external flow) defining the computational flow domain geometry, where flow will occur and the boundary domains with solid structure are crucial for the correct application of boundary conditions. Based on the requirements of the problem, geometry can be drawn in two dimensional (2-d) or three dimensional (3-d) planes. Using the clip planes across the geometry that reflect symmetry and/or rounding the sharp corners simplify the geometry. These actions facilitate the computational effort and solution accuracy while applying a precise mesh.

3.2.1.2 Mesh generation

Mesh generation is the second step of the pre-process stage. Applying the proper mesh demands the greatest consideration, because it concludes either success or failure in computing numerical solutions to the governing partial differential equations. An efficient mesh not only removes the problems that can lead to solution instability or lack of convergence, but also increases the likelihood of attaining the eventual solution of a CFD problem. The criterion for a well-constructed mesh is subdividing the whole domain geometry into numerous smaller domains such that these subdomains do not overlay on each other. The discretized computational domain must adequately resolve the important discrete values of the flow properties including velocity, pressure and temperature, and capture all the geometric details within the flow region.

Mesh topology consists of four categories, which are structure mesh, unstructured mesh, multi-block meshes and hybrid meshes. Structured and unstructured meshes are the most commonly applied methods. Multi-block and hybrid meshing are selectively used in extremely complex geometries. The structured mesh is the most straightforward approach to employ an orthogonal grid. In this method, the grid lines follow the coordinate directions to fit a geometric body. Body-fitted grid allows non-orthogonal and deformed cells. The cell surfaces within this grid layout are permitted to follow the surface of the
domain boundaries. **Figure 3.2** represents a body-fitted structured mesh. The application of the structured mesh in any aspect of grid generation has certain advantages and disadvantages. The structured mesh enables describing the elemental cells by indices (i; j; k). Additionally, the connectivity between the cell faces is straightforward; thus, data management and programming can be managed easily as compared to the unstructured mesh. On the other hand, this method has limited capabilities on complex geometries. Nonorthogonality and skewness generate unphysical results due to lack of success in transforming governing equations from one cell to the other. That impacts the accuracy and the numerical efficiency.

![Structured body-fitted mesh system.](image)

**Figure 3.2:** Structured body-fitted mesh system.

The unstructured meshing technique is typically employed to freely assemble within the computational domain. Generally, the unstructured mesh comprises cells in shape of triangle (2-d) and tetrahedron (3-d). Nevertheless, any other elemental shape including quadrilateral or hexahedral cells is possible as well. **Figure 3.3** shows an unstructured meshed geometry. This type of meshing performs sufficiently in complex geometries and domains with high curvature boundaries. Downsides associated to the unstructured mesh are related to; increased computational times due to data treatment and connectivity of elemental cells to transport more complex solution algorithms, and ineffective resolution at the wall boundary layers. Triangular and tetrahedral cells are not as prospering as quadrilateral or hexahedral cells due to long thin triangular cells at the walls bringing out major problems in approximating the diffusive fluxes.
Common element shapes utilized in the meshing process are in shape of triangle and quadrilateral (2-d) and tetrahedral, hexahedral, pyramids and prisms (3-d). Unlike structured and unstructured mesh systems, the hybrid meshing method combines more than one kind of element model to match mesh elements with the boundary surfaces, and assigns cells of diverse types in other parts of the complex flow regions. The intention is to improve the grid quality through the placement of quadrilateral and hexahedral cells near boundary layers, and the placement of triangle and tetrahedral cells for the rest of the flow domain. Herewith, the near wall regions are well-constructed and bulk area is meshed sufficient enough to provide decent mesh quality. The hybrid meshes can be non-conformal with grids that do not match at block boundaries and this capability allows replacing some portion of the mesh being changed.

The accuracy of a CFD solution is solely regulated by the presence of well-distributed sufficiently fine cells. As the size of individual cells is decreased, the number of cells in the mesh increases. Consequently, accurate solutions are led by the large number of elements and nodes; yet, calculation time and the computational capability requirements associated with increased number of elements and nodes are restrictions. Some practical guidelines while generating a well-performing mesh are listed as follows.

- For the same cell count, hexahedral meshes tend to give more accurate solutions, especially if the grid lines are aligned with the flow.
- The mesh density should be high enough to capture all relevant flow features especially for fluid flows having high-shear and/or high temperature gradients.
- The grid cells adjacent to the wall should form a fine mesh to resolve the boundary layer
flow.

- In boundary layers, quadrilateral, hexahedral and prism cells are preferred over triangular, tetrahedral or pyramid cells.

As aspect ratio, skewness, warp angle, and smoothness are the most populated mesh metrics to measure the grid quality. These factors are monitored by the cell shape and/or the distribution of cells. Aspect ratio is the ratio of mesh spacing in different directions. In Figure 3.4, the aspect ratio is mesh spacing on y-direction ($\Delta y$) divided by mesh spacing on x-direction ($\Delta x$). Large aspect ratios should always be avoided in important flow regions (e.g. interior flow domain) due to possible reduction in accuracy and poor iterative convergence. Maintaining a typical aspect ratio (AR) range of $0.2 < AR < 0.5$ within the interior region is a suggested practice. The skewness is related to the angle $\theta$ between grid lines as indicated in Figure 3.4. Typically angle $\theta$ should be approximately 90 degrees (hex and quad grids). As the angle $\theta$ becomes less than 45 degrees or greater than 135 degrees, the mesh consists of highly skewed cells, which can potentially lead to unrealistic results or numerical instabilities. The angle between the grid lines and the boundary of the computational domain (e.g. wall, inlet, or outlet) should be as close to 90 degrees as possible. Range of skewness is defined between 0 and 1 such that:

- 0 to 0.25 is excellent cell quality,
- 0.25 to 0.5 is good cell quality,
- 0.5 to 0.8 is acceptable cell quality,
- 0.8 to 1 is poor cell quality.

In case an unstructured mesh is adopted, warp angle also plays a significant role. Warp angle concerns the angle between the surfaces normal to the triangular parts of the faces and it should not be greater than 75 degrees due to serious convergence issues. As the cells have warp angles greater than 75 degrees, the problem can be overcome by a grid-smoothing algorithm. Smoothness is attained by the sequential expansion rate of adjacent cells. The variations in cell size should not be sudden. If the cells fail to have a gradual expansion in size, the truncation errors arise from discontinuities in grid size, and
these errors usually contain diffusive terms, where the discretization imposed require small changes. Ideally, the maximum change in grid spacing should not exceed 20%.

Preferably, a grid independence study can be performed to verify the suitability of the mesh applied and to yield an estimate of the numerical errors in the simulation for each class of problem. A typical method applied on grid independence study employs different grid resolutions by doubling the grid size in each direction. Then findings are extrapolated. If this is not feasible, some advanced methods including selective local refinement of the grid in critical flow regions of the domain can be applied, or comparing different order of spatial discretization on the same mesh may be applied. The grid independence study for this project is available in the CFD model design section.

![Figure 3.4: A quadrilateral cell with mesh spacing of \( \Delta x \), \( \Delta y \) and angle \( \theta \).](image)

3.2.1.3 Material properties

CFD can be applied to variety of problems ranging from radiation to compressible flow. In order to define the fluid flow characteristics properly, careful identification of the underlying flow physics for the fluid flow system to be modeled is of interest. CFD offers viscous, laminar, incompressible and isothermal flow options as flow physics, density and viscosity (dynamic) as fluid properties, and thermal conductivity and specific heat as heat transfer options. Figure 3.5 lists these properties.
3.2.1.4 Boundary conditions

Specifying the accepted boundary conditions follows material property selection as the fourth step. Boundary conditions restrict the simulation in a way to represent the real physical setting of fluid flow into a solvable CFD problem. While simulating the flow in a pipe, the boundary conditions are applied in different portions of the geometry. The inflow boundaries accommodate the fluid behavior entering the fluid domain; whereas, the outflow boundaries monitor the fluid leaving the domain. The interaction between the inlet and the outlet boundaries can be in the form of coupled velocity inlet and pressure outlet or coupled pressure and mass flow rate; such that the inflow demonstrates mass flow source and the outflow demonstrates sink of the solution zone. In application, the outlet boundaries are set further than the downstream not to alter the interior solution. The wall boundaries frame the flow geometry and set surrounding walls of flow domain. By default, viscous fluid flow between two parallel walls is characterized with no slip boundary conditioned flow. This condition states that fluid velocity comes to rest at the walls and peaks in the center of the clearance. In the presence of moving walls,
boundary conditions should reflect the precise physical conditions of the problem, such as tangential velocity, magnitude, and direction. As the problem gets more complex, the dynamic mesh technique can be implemented.

### 3.2.2 CFD Solver

The particular solution process in CFD consists of:

- Solution initialization,
- Solution control,
- Monitoring solution,
- CFD calculation,
- Convergence checking.

In order to resolve the problems, CFD utilizes the finite volume method with specified under-relaxation factors that determine the size of steps taken in iterative solutions for governing equations.

#### 3.2.2.1 Solution initialization

The iterative solution procedure of CFD requires an initial set of values including velocity, pressure and temperature to calculate the solution. When the initial conditions do not reflect the actual situation, the solution process tends to yield greater computational effort and the lack of convergence. Once the solution is initialized, it is controlled by the interpolation schemes and iterative solvers. First order upwind, second-order upwind, second-order central and quadratic upstream interpolation convective kinetics (QUICK) are the most common discretization structures. These methods are selectively applied based on the case specifications. Upwind and QUICK methods use interpolation with respect to flow direction.
3.2.2 Convergence monitoring

The solution process is finalized by checking for convergence on residual trends plotted during the solution process. These residuals can demonstrate progress of continuity equation, momentum equation in x,y,z, direction and turbulence parameters. Attaining the proper under-relaxation factors can significantly assist to speed up the convergence progress. The residual curves converge as they satisfy the specified tolerance values that stand as convergence criteria. In a broad view of a CFD project, convergence, convergence criteria or tolerance values attained for diverse equations, residuals, stability, under-relaxation factors, and grid independence cooperate to successively manage iterative process.

3.2.3 Post-Processing

Post-processing work of a CFD model includes reporting findings and visualizing the case. This final step of the process is significant to thoroughly describe the solution either in a local scale or in a global view. Depending on the type of information sought, X-Y plots, vector plots and contour plots can be employed. X-Y plots distribute the two findings in an x-y Cartesian system in quantities. Usually the variation of one term denoted in one axis control the other term. These two-dimensional plots enable extracting precise data almost everywhere in the fluid domain. This is done by drawing a line in the fluid domain and sourcing this data set in X-Y plots. The data set generated beyond the X-Y plot curve can be exported to any other software to process. On the other hand, the contour plots commonly serve to illustrate graphical representation of data. These plots tend to show global description of fluid flow in captured view and they are effective to identify trend changes in the flow region. Another way of presenting the results is vector plots whereby a vector quantity is displayed by the intensity of arrow distribution. X-Y plots and contour plots are particularly shown in the next chapter as the results are interpreted.
3.3  CFD Model Design

The annular hydraulics model demonstrated in this thesis presents a theoretical study conducted through computational fluid dynamics (CFD) technique to simulate drilling fluid flow. Specifically it identifies the velocity and pressure profiles in a typical annulus of CwD setting (a high diameter ratio, eccentric annulus and rotating inner pipe) with certain assumptions. Furthermore, the focus is on evaluating the influence of crucial parameters (eccentricity and presence of inner pipe rotation) on these profiles. The main advantage of using CFD in this work results from the feasibility of simulating different cases, verifying with the available numerical and analytical approaches, and operability to change parameters. The results and interpretation will be presented in the following chapter.

3.3.1 Geometry of the CFD Model

ANSYS Fluent offers several options to create geometry. By using the workbench design modeler geometry step, flow domains and walls are constructed. Specifically for this geometry, circles with a given origin are drawn on designated plane in 2-d. Extruding these circles along y-axis creates body; the outer wall stands for wellbore and the inner wall stands for casing. This extrusion embodies the annular section so that the wellbore encapsulates the casing. (The diameter of casing is designed as 7 inches and the diameter of wellbore is given as 8.75 inches. This satisfies a pipe diameter to well diameter ratio of 0.8. These diameter values and the ratio is an applicable data set for practical production zone drilling and reside in the applicable range defined by several authors (Lopez et al. (2010), Fontenot et al., (2004), Rosenberg et al., (2010), Torsvoll et al., (2010), Karimi et al., (2011))). For geometric model construction of the concentric cases, origin-centered circles are drawn. For the eccentric cases, the plane, on which the inner pipe resides, is shifted to yield the desired eccentricity. The coordinates are determined by the formulation below assuming the circles are drawn in a 2-d x-y plane. These formulae help to attain how much to shift the plane has to the expected eccentricity. Figure 3.6 illustrates the terms mentioned in this formulation.
\[ \delta = \left( D_H - D_p \right) \frac{e}{2} \]  

(3.1)

where \( \delta \) (in.) is center-to-center distance, \( e \) is eccentricity and \( D \) (in.) is diameter.

\[ Coordinate + x = \left( \sqrt{\frac{D_p}{2}} \right)^2 - \delta^2 \]  

(3.2)

\[ Coordinate - x = \left( \sqrt{\frac{D_p}{2}} \right)^2 - \delta^2 \]  

(3.3)

\[ Coordinate + y = \frac{D_p}{2} + \delta \]  

(3.4)

\[ Coordinate - y = D_p - Coordinate + y \]  

(3.5)

Figure 3.6: Coordinates of an eccentric annulus in geometry modeler.

For the favor of computational time and effort, and mesh restrictions, pipe length is designed as six feet and the flow is considered to take place only in the annular section. The annular section mentioned here is designated as the fluid domain and bounded by four surfaces; namely, inlet velocity (for inflow), outlet pressure (for outflow), casing, and wellbore (walls).
In order to start calculations, a given data set is utilized. Using these, further calculations are conducted and the final input parameter table is drawn. To start with, the following parameters are set as shown in Table 3.1.

Table 3.1: Set of data for input parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole Diameter, $D_H$</td>
<td>8.75</td>
<td>in.</td>
</tr>
<tr>
<td>Pipe Diameter, $D_P$</td>
<td>7</td>
<td>in.</td>
</tr>
<tr>
<td>Length of section, $L$</td>
<td>6</td>
<td>ft</td>
</tr>
<tr>
<td>Casing rotational speed, $N$</td>
<td>0</td>
<td>rpm</td>
</tr>
<tr>
<td></td>
<td>80</td>
<td>rpm</td>
</tr>
<tr>
<td>Eccentricity, $e$</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.6</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>Fluid Density, $MW$</td>
<td>9.8</td>
<td>ppg</td>
</tr>
<tr>
<td>Fann Reading @ 600 rpm</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td>Fann Reading @ 300 rpm</td>
<td>23</td>
<td></td>
</tr>
<tr>
<td>Flow Rate, $Q$</td>
<td>250</td>
<td>gpm</td>
</tr>
</tbody>
</table>

Input parameters for the model are finalized through several calculations. In these rheology calculations, the terms and formulation excerpted from SPE’s method for Power Law Fluids are applied (Bourgoyn et al., 1986).

Having the flow rate and diameters, mean fluid velocity is given as:

$$v = \frac{0.408Q}{D_H^2 - D_P^2} \quad (3.6)$$

where $D$ (in.) is diameter, $Q$ (gpm) is flow rate, $v$ (ft/sec) is fluid velocity.

Rheological properties of Power law fluids are described by the formulation below:

$$n = 3.32 \log \left( \frac{\theta_{600}}{\theta_{300}} \right) \quad (3.7)$$

$$K = \frac{511\theta_{300}}{511^n} \text{ (eq. cp)} \quad (3.8)$$
Reynolds Number in annulus is found by the following calculation

\[ N_{Re_a} = \left( \frac{109100 \times MW \times v^{2-n}}{K} \right) \times \left( \frac{0.02098 \times (D_H - D_P)}{2 + \frac{1}{n}} \right)^n \]  

(3.11)

In this formulation, MW (ppg) is fluid density, \( v \) (ft/sec) is fluid velocity, \( K \) (eq cp) is consistency index, \( n \) is Power Law index, \( D \) (in.) is diameter.

The flow regime is simply determined by comparing Reynolds number of given conditions with the critical Reynolds number. The critical Reynolds number is determined from a friction factor vs. Reynolds number chart with corresponding \( n \) numbers. This chart is shown in Figure 3.7. Friction factor \( f \) equation for power law fluids equation is given by:

\[ \sqrt{f} = \frac{4}{n^{0.75}} \log \left( N_{Re} \left( \frac{n}{1.2} \right)^{1-n} \right) - \frac{0.395}{n^{1.2}} \]  

(3.12)

Figure 3.7: Friction factors for Power Law Fluids (Bourgoyne et al., 1986).
With the conditions listed above, friction factor (f) is found to be 0.01026 and Power law index (n) is found to be 0.605. These values correspond approximately to the critical Reynolds number of 2000. For the specified flow rate of 250 gpm, Reynolds number calculations are performed and it is found to be 1511. As a result, the laminar flow is secured. Rotating inner pipe case is known to introduce Taylor’s number and alter the Reynolds number positively. Additionally, the eccentricity influences Reynolds number distribution. Nevertheless, these fluctuations in Reynolds number are not included in this example to maintain laminar flow. The rotational speed that inner pipe possesses in this model are lower than a certain level of revolution per minute. The alterations on viscosity due to rotating inner cylinder are mainly associated to cases with high centrifugal actions. On the other hand, regarding the eccentricity effect, shear rate dependent viscosity model as accepted and applied by ANSYS Fluent, already considers contribution of eccentricity intuitively. The viscous flow modeling condition in the model includes the shear rate dependency.

Depending on the cases and initial conditions, these values and equations set can be varied. However, this example of data set and formulations will be sufficient to demonstrate the model developed in this thesis.

The final form of input parameters for each step (eccentric, concentric, inner pipe rotating and inner pipe stationary) is shown in Table 3.2. For illustration purposes, only 0.5 eccentricity case is revealed here to represent eccentric geometries. Yet, specific data for each case was inputted in the model. It is essential to note that this data set is not obtained from any field result, only concern while adapting this data set is to simulate realistic CwD conditions.

### 3.3.2 Meshing of the CFD Model

By using the workbench design modeler meshing step, a mesh system is created over the geometry constructed in the initial step. ANSYS Fluent lets the user control mesh density, cell size, cell distribution in local and global parts and cell shapes. For this specific study, the sweep method is applied
along with face sizing. This approach enables the definition of cell distribution on the inlet face, and to sweep it across the fluid domain. The main purpose in doing so is to carefully define cell distribution on critical regions. For the annular fluid flow in a vertical well, most of the critical alterations are observed on the inlet surface in horizontal plane, especially in eccentric geometries. It is of great importance to capture every single detail with a selective mesh in this region. Once the flow is in progress on the fluid domain, the iterations get simpler to follow each other. In the flow domains simulated in this study, the simulation process does not meet immense difficulties, because the geometry does not have any geometric irregularities in flow direction and this helps in transporting solution between the neighboring cells in axial direction. With this procedure, while the accuracy of the solution is preserved, computational effort and time are optimized. Another technique that was employed to capture details, is the mesh system controlled by quadrilateral elements. The quadrilateral cells are convenient for this case, because there are no triangle elements that can distort the grid. While various other terms including smoothing, inflation, growth rate and transition are applied as program controlled default values, mesh sizing is introduced to achieve a solution that is not dependent on grid. Upon completing a comprehensive grid independence study, the optimum size of grids are defined in a range between 1.5 mm to 3 mm. Depending on the geometry (eccentricity) specifications, the, mesh systems include between 100,000 to 200,000 cells with this sizing. Furthermore, the mesh quality was clinched by mesh metrics (Skewness<0.05 and AR≈0.25). Figure 3.8 presents a sample meshed geometry excerpted from 0.4 eccentric case.

A grid independence study verifies that the solution is constructed on a mesh model that effectively facilitates the numerical solution. Commonly, three or four different cell sizes are examined and the error percentage is compared to each other. The findings are extrapolated and the optimum configuration is attained considering the limitation on number of elements, computational time and effort, and solution accuracy. The main principle is to double the size of elements in each try. This provides a global look on the mesh precision. Since the mesh sizing is defined in a particular range during the model
Table 3.2: Finalized input parameters for concentric and 0.5 eccentric case.

<table>
<thead>
<tr>
<th>SCENARIO</th>
<th>SCENARIO</th>
<th>SCENARIO</th>
<th>SCENARIO</th>
</tr>
</thead>
<tbody>
<tr>
<td>GEOMETRY</td>
<td>concentric, w/o rotation, laminar</td>
<td>50 % eccentric, w/o rotation, laminar</td>
<td>concentric, with rotation, laminar</td>
</tr>
<tr>
<td>Hole Diameter (D_h)</td>
<td>8.75 in 0.222 m</td>
<td>8.75 In 0.222 m</td>
<td>8.75 in 0.222 m</td>
</tr>
<tr>
<td>Pipe Diameter (D_p)</td>
<td>7 in 0.178 m</td>
<td>7 in 0.178 m</td>
<td>7 in 0.178 m</td>
</tr>
<tr>
<td>Length of Casing (L)</td>
<td>6 ft 72 in</td>
<td>6 Ft 72 in</td>
<td>6 ft 72 in</td>
</tr>
<tr>
<td>Casing Rotation</td>
<td>0 rpm 0 rad/s</td>
<td>0 rpm 0 rad/s</td>
<td>80 rpm 8.378 rad/s</td>
</tr>
<tr>
<td>ECCENTRICITY</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Eccentricity (e)</td>
<td>0</td>
<td>0.5</td>
<td>0</td>
</tr>
<tr>
<td>Center to Center Distance (δ)</td>
<td>0 in 0 m 0.4375</td>
<td>0 in 0 m 0.4375</td>
<td>0 in 0 m 0.4375</td>
</tr>
<tr>
<td>Coordinate +x</td>
<td>3.5 in 0.0889 m 3.473</td>
<td>3.5 in 0.0889 m 3.473</td>
<td>3.5 in 0.0889 m 3.473</td>
</tr>
<tr>
<td>Coordinate -x</td>
<td>3.5 in 0.0889 m 3.473</td>
<td>3.5 in 0.0889 m 3.473</td>
<td>3.5 in 0.0889 m 3.473</td>
</tr>
<tr>
<td>Coordinate +y</td>
<td>3.5 in 0.0889 m 3.938</td>
<td>3.5 in 0.0889 m 3.938</td>
<td>3.5 in 0.0889 m 3.938</td>
</tr>
<tr>
<td>Coordinate -y</td>
<td>3.5 in 0.0889 m 3.062</td>
<td>3.5 in 0.0889 m 3.062</td>
<td>3.5 in 0.0889 m 3.062</td>
</tr>
<tr>
<td>FLUID PROPERTIES</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drilling Fluid Density (MW)</td>
<td>73.3 lb/ft^3 9.80 ppg</td>
<td>73.3 lb/ft^3 9.80 ppg</td>
<td>73.3 lb/ft^3 9.80 ppg</td>
</tr>
<tr>
<td>Drilling Fluid Velocity (v)</td>
<td>3.70 ft/s 1.128 m/s</td>
<td>3.70 ft/s 1.128 m/s</td>
<td>3.70 ft/s 1.13 m/s</td>
</tr>
<tr>
<td>Fann Reading @ 600 rpm</td>
<td>35</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>Fann Reading @ 300 rpm</td>
<td>23</td>
<td>23</td>
<td>23</td>
</tr>
<tr>
<td>Drilling Fluid Viscosity (µ)</td>
<td>17.5 cp 0.0118 lb/(ft-s)</td>
<td>17.5 Cp 0.0118 lb/(ft-s)</td>
<td>17.5 cp 0.0118 lb/(ft-s)</td>
</tr>
<tr>
<td>n</td>
<td>0.605</td>
<td>0.605</td>
<td>0.605</td>
</tr>
<tr>
<td>K</td>
<td>0.73 lb ft^2/100 ft^2 0.27 kg s^2/(ms^2)</td>
<td>0.73 lb ft/100 ft^2 0.27 kg s^2/(ms^2)</td>
<td>0.73 lb ft/100 ft^2 0.27 kg s^2/(ms^2)</td>
</tr>
<tr>
<td>Flow Rate</td>
<td>250 gpm</td>
<td>250 gpm</td>
<td>250 gpm</td>
</tr>
<tr>
<td>N_R</td>
<td>1511</td>
<td>1511</td>
<td>1511</td>
</tr>
</tbody>
</table>
Figure 3.8 Preview of 0.4 eccentric geometry

generation step, the minimum and maximum ends of range are doubled in each try. Four different cell sizes are investigated and they are denoted as A0, A1, A2 and A3. Table 3.3, Figure 3.9 and Figure 3.10 summarize the grid independence study conducted for the 0.5 eccentric non-rotating inner pipe scenario. This verification method is not only applied to 0.5 eccentric case, however the most obvious results were obtained in this case. While error comparison chart is being drawn, the pressure loss gradients obtained by the simulation of different mesh configurations are compared to Haciislamoglu’s (1990) pressure loss equation for eccentric geometries. Haciislamoglu’s equation to calculate pressure loss is given in the previous chapter. The error is calculated with the equation as shown:

$$Error = \frac{(\frac{dP}{dy})_{CFD} - (\frac{dP}{dy})_{Numerical}}{(\frac{dP}{dy})_{Numerical}} * 100$$

(3.13)
Table 3.3: Description of the mesh configurations used in grid independence study.

<table>
<thead>
<tr>
<th>Model Notation</th>
<th>A0</th>
<th>A1</th>
<th>A2</th>
<th>A3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum cell size</td>
<td>0.0008m</td>
<td>0.0015m</td>
<td>0.003m</td>
<td>0.006m</td>
</tr>
<tr>
<td>Maximum cell size</td>
<td>0.0015m</td>
<td>0.003m</td>
<td>0.006m</td>
<td>0.012m</td>
</tr>
<tr>
<td>Number of nodes</td>
<td>335895</td>
<td>161832</td>
<td>40215</td>
<td>10983</td>
</tr>
<tr>
<td>Number of elements</td>
<td>307820</td>
<td>145808</td>
<td>34160</td>
<td>8420</td>
</tr>
<tr>
<td>Skewness criteria</td>
<td>90% of elements less than 0.05</td>
<td>90% of elements less than 0.05</td>
<td>Distributed between 0.01 and 0.15 with 70% of elements less than 0.05; 20% of elements between 0.05 to 0.1; 10% of the elements between 0.10 to 0.15</td>
<td>Distributed between 0.01 and 0.4 with 25% of elements less than 0.05; 30% of elements between 0.05 to 0.1; 20% of the elements between 0.1 to 0.2; 15% of the elements between 0.2 to 0.3; 10% of the elements between 0.3 to 0.4</td>
</tr>
<tr>
<td>Pressure loss gradient</td>
<td>0.028483 psf/ft.</td>
<td>0.028417 psf/ft.</td>
<td>0.029267 psf/ft.</td>
<td>0.02895 psf/ft.</td>
</tr>
<tr>
<td>Error</td>
<td>1.187028%</td>
<td>1.418304%</td>
<td>1.530474%</td>
<td>0.431909%</td>
</tr>
<tr>
<td>Comments</td>
<td>Excessive computational time and effort</td>
<td>Moderate computational time and effort with an approximate error percentage to model A0</td>
<td>Fairly less computational time and effort but it fails to refine the grid system as seen in error percentage and skewness criteria</td>
<td>Minimized computational time and effort but it significantly lacks accuracy due to skewness criteria</td>
</tr>
</tbody>
</table>

As shown is the Table 3.3, the cell sizes are altered in a doubling trend. To recall, cell size in the grid system is defined by a range. So in Table 3.3, minimum and maximum sizes show the range of cell size in the denoted model. In the A0 model, the finest cells are created with a range of 0.0008 m to 0.0015 m. This range is doubled for the following mesh model and the series continue with this trend. By this way, the refinement of the grid systems is loosened and the results for each are observed. Once the elements sizes in the grid system are altered, the counts of nodes and elements that are contained in the flow domain are redefined. As it is observed, increasing element size reduces the number of elements in the grid system and the number of nodes where those cells are in contact with each other. Another aspect that changes with various cell sizes, is the skewness criteria. The skewness criterion has been described earlier in this chapter in the mesh generation section. To recall, a skewness range of 0 to 0.25 stands for an excellent quality mesh system. In the Table 3.3, mesh accuracy based on skewness is shown. The skewness criteria row answers the question “What percentage of the whole cells fall into which skewness values?” To illustrate, in the model A2, 70% of all elements in the fluid domain (34160 elements) have a skewness of less than 0.05, 20% of all elements have skewness ranged between 0.05 and 0.1, and the skewness for the rest of the cells are ranged between 0.10 and 0.15. Regarding the excellence range is between 0 and 0.25, this mesh model satisfies. However, as the cell sizes are reduced (models A0 and
A1), the skewness quality improves even more now that more elements fall into the region where skewness is less than 0.05. The pressure loss gradient and error rows in Table 3.3 show how close the solutions are to the numerical results obtained by Eq. 2.13. The pressure loss gradient demonstrates the exact value that the designated model yielded. For instance, model A1 yielded 0.028417 psi pressure loss over a vertical interval of one foot. This result has a deviation of 1.42 % from the result calculated by the numerical solution (0.028825). The proximity of the model result to the numerical result is attained by the Eq. 3.13. As the mesh refinement enhances, the proximity to actual result increases. After all these discussions, an interpretation on the Table 3.3 yields that the mesh model created in model A0 shows the finest mesh system and shows a correlation with A2 and A1, with an increasing number of elements, increasing accuracy and decreasing error. However, the excessive computational process as compared to A1 puts this configuration into an unfeasible category. In that level of refinement, keeping the computational time for too short a time results in distorted values and the solution does not converge. Yet, the improvement on error percentage is not significant enough to warrant this computational effort. On the other hand, A3 stands as the best solution by looking at the errors. However, this is more related to oversimplification carried by coarse mesh. It is a coincidence because the general principle is that the finer the mesh is, the more precise the solution is. This opinion is supported by the poor skewness distribution shown in the skewness criteria row. Considering all these facts, the A1 model is selected for this case. Likewise, this sizing range performs well for the rest of the cases. It should be noted that a sweep method is employed; thus, varying element sizing will only affect the cell distribution on the source surface. Following that, all the distribution will be swept through the body until it hits the target surface. Therefore, playing with cell size does not necessarily affect all dimensions of the cells. Distance of cells to each other (sweep element size) on y direction (sweep direction) stays constant.

It is confirmed that the solution is neither grid dependent nor convergence dependent. Convergence dependence is checked; and the optimized convergence is caught with residual tolerance of $1 \times 10^{-4}$.  

52
Figure 3.9: Error comparison of different grid configurations.

Figure 3.10: Pressure drop curve for 0.5 eccentric annular with different mesh configurations.
3.3.3 Assumptions and Boundary Conditions

CFD offers considerable flexibility to vary many parameters including diameter ratio, rheological properties of power law drilling fluid, eccentricity, flow rate, fluid density, and casing rotational speed. Though, this project is personalized to only comprise the effect of eccentricity and the effect of presence of a given rotational speed (80 rpm). The main purpose beyond these assumptions is to simulate a realistic CwD annular environment. Besides, flow rate and eccentricity are set such that laminar flow regime is sustained in the fluid domain. With these settings, it is intended to investigate sixteen different scenarios. ANSYS Fluent is capable of computing annular pressure and velocity profile for the scenarios is listed in Table 3.4.

Table 3.4: Cases simulated in this dissertation.

<table>
<thead>
<tr>
<th>Flow Type</th>
<th>Inner Pipe Rotation</th>
<th>Annulus Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar Flow</td>
<td>Without rotation</td>
<td>Concentric</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>30% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>40% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>50% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>60% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80% eccentricity</td>
</tr>
<tr>
<td></td>
<td>With rotation (80 rpm)</td>
<td>Concentric</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>30% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>40% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>50% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>60% eccentricity</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80% eccentricity</td>
</tr>
</tbody>
</table>

As summarized in Table 3.4, power law fluid flow through the concentric and the eccentric annulus with and without rotation are introduced. In the literature review, the complexity and variation of mathematical models beyond the inner pipe rotating cases are described. To recall, the shear rate
distribution in annuli becomes a function of tangential, horizontal and axial motion, consequently the numerical solutions to the governing equations do change. The flow of power law fluid in the eccentric borehole annuli with rotation distorts from the expected trends, because it exhibits shear-dependent changes to viscosity and this applies to the pressure gradient, which depends on rotational speed, fluid density and apparent viscosity.

For the sake of practicality, a number of assumptions are introduced. The assumptions stated for this project are listed as:

1. **Uniformly eccentric and concentric geometry:** The wellbore geometry presented in the model has a uniform distribution of eccentricity throughout the well. Eccentricity will not change by depth.

2. **Power Law Fluid:** Commonly encountered drilling fluids are either Herschel-Bulkley or power law fluids. In the research projects, power law fluids are generally preferred over Herschel-Bulkley fluids due to its simplicity in mathematical representation and computing processes. Power law model drilling fluid is of interest in this model with the power law index (n) < 1. Rheological properties (n, K, viscosity) of the fluid are kept constant during the process. Thixotropic nature of drilling fluid is ignored.

3. **Laminar Flow:** Regardless of the eccentricity and inner pipe rotation, fluctuations in Reynolds number are assumed to be negligible and the flow regime is kept in laminar form.

4. **Constant rotational speed:** In inner pipe rotating cases, the rotational speed is set to 80 rpm and the inner pipe is globally rotating (not locally rotating parts).

5. **Incompressible fluid:** Drilling fluid density alteration over time and distance are not considered. Drilling fluid is assumed to preserve its original properties. Density fluctuations from the wellbore fluid intrusion and cutting concentration are ignored.

6. **Isothermal conditions:** Temperature is not included in any section of this model.

7. **Vertical well:** As CwD is used to pass through troublesome zones; it makes sense that these zones are encountered in the vertical section, because the horizontal section mostly resides in the
homogenous reservoir zone. Also, convenience of the CwD drill string geometry favors vertical well drilling.

8. **3-d model with main flow in axial direction.** The model created is a 3-d model and the flow acts in the axial direction. By rotating inner pipe, tangential flow does form but the main flow is always governed by the axial flow. Earth’s gravitational factor of 32.174 ft/s² acts against the flow.

9. **Steady state and fully developed flow:** The flow is not time dependent. Also, for the sake of observing the accurate pressure gradient in any depth of annulus, entrance effect (hydraulic length) has been eliminated by modifying velocity components at the inlet.

10. **Non-retrievable casing drilling system:** For the sake of simplicity in drill string geometry and to utilize high rotational speed of casing string, the non-retrievable CwD system is preferred.

11. **Pipe diameter to hole diameter ratio:** The ratio of about 0.8 is considered. It is secured in a uniform borehole with constant diameter by ignoring cutting accumulation on the wellbore. Sections of open hole are circular in shape and known diameter. Particle size distribution is not included in any part of the model.

Boundary conditions of this model vary depending on the existence of inner pipe rotation. Regardless of the scenario, the fluid is in contact with the walls and abides to them. The outer wall as a boundary condition is always kept stationary. For the non-rotating inner pipe case, the inner wall is stationary as well. Since the viscous fluid flow is in action, no slip boundary conditions control the wall boundary surfaces. For the rotating inner pipe case, rotational pipe motion (80 rpm) is prescribed at the inner surface. Inlet and outlet boundaries are coupled with inlet velocity of 3.70 ft/s and atmospheric outlet pressure.

### 3.3.4 Model Validation

Model validation provides the insight about the correct track followed in the modeling process. The model validation efforts are conducted through annular frictional pressure loss comparison based on
the presence of the inner pipe rotation. As the pipe is non-rotating, the axial momentum equation characterizes the flow. It is a steady-state solution procedure and can be calculated by applying the prescribed equations available in the literature. Instead, as the inner pipe rotation and the eccentricity combination are taken into account, azimuthal momentum additionally acts on the mathematical models and analytical solutions turn into tangled second order partial differential equations, which exceeds over soluble by hand range in exact forms. These are linked to the altered shear rate, which is led by velocity profiles with the introduction of tangential movement on top of axial movement of mud. They were expressed previously under the “Application of Computational Fluid Dynamics in Drilling Engineering” section on the account of describing the necessity to use CFD in this project. Consequently, validating each scenario is not theoretically possible due to lack of manual solutions for each case. Instead, pressure loss and velocity could be calculated for the following scenarios; concentric-without rotation-laminar flow and varying eccentricity (0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.8) -without rotation-laminar flow. These results are regarded as reference points and the pressure loss gradient is calculated using equations derived by Uner et al. (1988) and Haciislamoglu et al. (1990). The incentive beyond this is to compare solution created through CFD (utilizing finite volume technique) with the solution created through Haciislamoglu’s numerical model (utilizing finite differences technique) and with Uner et al.’s analytical model (utilizing narrow slot approximation technique). In the rotating inner pipe case, the deviations from these validated values are observed. Overall, the study showed that with increasing eccentricity, pressure loss gradient decreases, and with the presence of inner pipe rotation, pressure loss gradient increases. Fictional pressure loss gradient results are tabulated in Table 3.5 and illustrated in Figure 3.11.

The results shown below are calculated following the formulae listed.

Uner et al. defined flow rate for concentric case as

\[
q_c = \frac{\pi r_0^3}{2} \frac{n}{2n+1} \left[\frac{dP}{dy}\frac{r_0}{2K}\right]^\frac{1}{n} (1 + R_r)(1 - R_r)^{\left(\frac{2+1}{n}\right)}
\]

(3.14)
Flow rate for eccentric case is given as

\[ q_e = \frac{\pi r_o^3}{2} \frac{n}{2n+1} \frac{1}{E} \frac{1}{2} \frac{1}{(2E-\pi R_r)^n} (1-R_r^2)^{1-n} F(f, n, R_r) \]  

(3.15)

where F, E, f and R, constants are given as

\[ F(f, n, R_r) = \int_0^\pi \left( \sqrt{1 - f^2 \sin^2 v + f \cos v - R_r} \right)^{2 + \frac{1}{n}} dv \]  

(3.16)

\[ E = \int_0^\pi \left( \sqrt{1 - f^2 \sin^2 v} \right) dv \]  

(3.17)

\[ f = \delta_r \frac{1-R_r}{R_o-R_i} \]  

(3.18)

\[ R_r = \frac{r_i}{r_o} \]  

(3.19)

\[ \delta_r = (r_o - r_i)e \]  

(3.20)

where, \( q_c \) (m³/s) is flowrate for concentric case, \( q_e \) (m³/s) is flowrate for eccentric case, \( r_o \) (m) is radius of outer wall of annulus, \( r_i \) (m) is radius of inner wall of annulus, K (Pa·s) is consistency index, n is Power Law index, P (Pa) is pressure, \( \delta_r \) (m) is center to center distance and \( R_r \) is radius ratio.

Haciislamoglu et al. equated ratio of pressure drop gradient in eccentric case to pressure drop gradient in concentric case as given below. This equation is dimensionless and requires minimal constraints.

\[ R = 1 - 0.072 \frac{e}{\pi} (R_r)^{0.8454} - 1.5e^2 \sqrt{n} (R_r)^{0.1852} + 0.96e^2 \sqrt{n} (R_r)^{0.2527} \]  

(3.21)

where,

\[ R = \frac{\frac{dP}{dy}_e}{\frac{dP}{dy}_c} \]  

(3.22)

where, \( \frac{dP}{dy}_e \) is pressure loss gradient for eccentric case, \( \frac{dP}{dy}_c \) is pressure loss gradient, \( R_r \) is radius ratio, \( r_o \) (m) is radius of outer wall of annulus, \( r_i \) (m) is radius of inner wall of annulus, n is Power Law index and e is eccentricity.
Table 3.5: Frictional pressure drop received in CFD, in numerical and analytical solutions.

<table>
<thead>
<tr>
<th>Eccentricity</th>
<th>Simulation</th>
<th>Haciislamoglu et al.</th>
<th>Error, %</th>
<th>Uner et al.</th>
<th>Error, %</th>
<th>Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0377</td>
<td>0.03722</td>
<td>0.03</td>
<td>0.039</td>
<td>0.93</td>
<td>0.04</td>
</tr>
<tr>
<td>0.1</td>
<td>0.03721</td>
<td>0.03722</td>
<td>0.03</td>
<td>0.03756</td>
<td>0.93</td>
<td>0.03814</td>
</tr>
<tr>
<td>0.2</td>
<td>0.03577</td>
<td>0.03576</td>
<td>0.03</td>
<td>0.03608</td>
<td>0.86</td>
<td>0.03835</td>
</tr>
<tr>
<td>0.3</td>
<td>0.03359</td>
<td>0.03376</td>
<td>0.50</td>
<td>0.03393</td>
<td>1.00</td>
<td>0.03854</td>
</tr>
<tr>
<td>0.4</td>
<td>0.03108</td>
<td>0.03141</td>
<td>1.05</td>
<td>0.03141</td>
<td>1.05</td>
<td>0.03868</td>
</tr>
<tr>
<td>0.5</td>
<td>0.02845</td>
<td>0.02883</td>
<td>1.32</td>
<td>0.02879</td>
<td>1.18</td>
<td>0.03803</td>
</tr>
<tr>
<td>0.6</td>
<td>0.02593</td>
<td>0.02623</td>
<td>1.14</td>
<td>0.02624</td>
<td>1.18</td>
<td>0.0349</td>
</tr>
<tr>
<td>0.8</td>
<td>0.02139</td>
<td>0.02151</td>
<td>0.56</td>
<td>0.02163</td>
<td>1.11</td>
<td>0.03447</td>
</tr>
</tbody>
</table>

Figure 3.11: Frictional pressure drop comparison for CFD, numerical and analytical solutions.
CHAPTER 4
INTERPRETATION OF CFD RESULTS

In this chapter, the results from CFD model are discussed, and the pressure and velocity variations are analyzed. All the results and different aspects of the cases are illustrated with different demonstration techniques available in the CFD post-processer. These techniques were introduced in the previous chapter. The model results are categorized according to their specific conditions. Starting from laminar flow with non-rotating inner pipe in the concentric annulus to laminar flow with rotating inner pipe in the highly eccentric annulus, the results are acquired. Rotating inner pipe and fixed inner pipe cases are individually treated. Particularly in this chapter, results for concentric, 0.3 eccentric, and 0.6 eccentric cases are interpreted. These cases will be evaluated through rotating and non-rotating inner pipe scenarios. Other results are available in Appendix A. The explanations in this chapter can help to elucidate the trends for the figures presented in the Appendix. The results are sought in the following fashions:

- Individual pressure contour plot in 3-d
- Individual velocity contour plot in 3-d
- Contrastive annular pressure vs. flow direction graph in 1-d
- Contrastive annular velocity across annulus graph in 1-d at same depth (at inlet)

With the tones of color, the contour plots represent either pressure or velocity distribution throughout the whole body (inner wall, outer wall, flow area, inlet surface area and outlet surface area). For the eccentric scenarios and especially cases with the dynamic inner pipe, they help to visualize the effects clearly. Considering that, multiple views of pressure contour plots from different orientations are captured. The pressure contour plots make it possible to calculate total pressure across the annulus by subtracting values in the colored label residing next to the plot. These plots show how gradually pressure decreases from inlet to outlet. For eccentric and rotating cases, pressure varies on the same horizontal
surface regarding resistance to fluid flow formed by geometry, contribution of rotational speed and tangential movement of pipe. For those cases, the weighted area integral of pressure is calculated on the surface of both ends of the pipe and average pressure values are calculated. On the other hand, for the velocity plot, the flow velocity profile in the annulus is captured. The plots confirm a narrow-slot, no slip boundary type flow model between two parallel plates; fluid velocity at the wall is minimized and maximized at the center.

1-d graphs of annular pressure represent the values of an imaginary line passing through the annular clearance. For the eccentric and inner pipe rotating cases, the pressure behavior in the annulus differs; therefore, all cases are described with two imaginary lines. These lines are designed to pass next to the wellbore and inner pipe so that the difference between them can be observed. They are notated as pressure drop near the wellbore and pressure drop near the casing. Although in fixed inner pipe cases, the pressure distribution is uniform across the same cross sectional area. For the sake of consistency and proving this statement, these cases are demonstrated with the same method. Specifically, these graphs explain the trend in the annular space from inlet to outlet. Using these, the effects of rotation, eccentricity, and flow regime can be interpreted.

1-d graphs of annular velocity represent the values of an imaginary line passing across the annular clearance in horizontal plane. For eccentric cases, velocity behavior in the annulus differs, therefore all cases are described through an imaginary line: passing perpendicular to the narrow side and the wide side. This approach reveals how diverse the results can be in the same surface due to severity of the eccentricity and the effect of rotation. Further interpretations of these graphs are available in the evaluation of the results section. Figure 4.1 shows the location of the imaginary lines that help to generate data on the sketch of sample geometry. The line residing at the bottom of the figure is used to generate data for X-Y plots showing annular velocity. Based on this data, the velocity profile graph on the same horizontal plane is drawn. The lines residing along the annulus are used to generate data for X-Y plots showing the pressure profile along the annulus. As discussed, rather than a single line, two lines
exist to observe the diverse profiles. The data generated on these lines is reflected on the annular pressure profile graph.

Figure 4.1: Lines passing through various sections of body to generate data for X-Y plots.

4.1 Effect of Casing Eccentricity in Stationary Pipe Conditions

String eccentricity is a crucial factor to consider while simulating the CwD annulus. Since the casing string is dynamic and locally under compression, any kind of eccentricity can be expected. Regarding that, a variety of eccentricity configurations have been run. All of the runs were coupled with a moderate flow rate of 250 gpm. This specific rate secures the laminar flow, thus the erosive forces of drilling fluid associated to turbulent flow are eliminated. This is the common application in the CwD operations in order not to damage the filter cake.

As the cross-sectional surface at any point of the wellbore is investigated, it is observed that the intrusion of the eccentricity leads flowrate to be unevenly distributed, as well as the velocity. Comparing
the figures of velocity contour plots in the following sections clearly present this. Along with the rheological model and exact annular geometry, the net volumetric flow rate is in connection with an applied pressure gradient and velocity profile. At instantaneous view, it varies with the position in the annular domain. While doing so, the main references are the boundary conditions and the well geometry for this model. The geometry governs such that the narrow side yields more resistance to flow. Hence, the flow will prefer to go to wide section on the same cross sectional area. This can also be explained by the stress level (confronting fluid flow) comparison of both sides. In order to verify the link to boundary conditions, especially to the inflow and outflow conditions, the physics beyond the flow must be well-understood. While the model is being set up in CFD, it is suggested that the outlet pressure is uniformly distributed on the cross-sectional outlet surface, which means that the pressure meeting the flow in narrow section of the annulus is same as the one in wide section. Since the outlet is bounded by a given pressure uniformly applied to cross section surface, starting from the inlet flow, CFD solves the problem in such a way that iterations imitate to downstream conditions, and keep solution progress consistent with the outlet by the time it reaches to outlet. For the sake of justification of this trend, the governing equation beyond the flow must be examined. Explicitly, at the flow domain, Bernoulli’s equation is solved to satisfy this condition and this situation redefines the velocity distribution. Here, 1-d incompressible steady state flow is considered, assuming the flow is in +y direction. Bernoulli’s equation for this condition is:

\[ P + \rho \frac{v^2}{2} = P_{total} \]  \hspace{1cm} (4.1)

where \( \rho \) is density, \( v \) is fluid velocity and \( P \) is pressure.

In this notation, dynamic pressure component of the total pressure is the term associated to \( v \)-velocity of fluid. The static pressure component for the case considered in this study is associated with the summation of the hydrostatic column of mud in annulus and pressure at outlet. The summation of these two terms describes total pressure (\( P_{total} \)). ANSYS Fluent user guide accepts Eq. 4.1 and the definitions associated to that for dynamic pressure, static pressure and total pressure.
According to the **Eq. 4.1**, the cross-sectional total pressure at the same depth as an eccentric annulus is expected to be different, because the velocity varies depending on the annular clearance in this study. However, this difference in total pressure would be observed in an open-ended media, not in an outflow bounded media. In this case, the boundary conditions limit the pressure outlet and turn the system into a closed system.

Utilizing the incompressibility of flow and steady state flow, pressure across a cross-section at the same depth of annulus balances each other such that a uniform pressure profile is distributed all over the surface of any cross sectional area at any depth (when it is viewed on the horizontal plane, the narrow section with tendency to resist flow enormously and the wide section with the majority of the flow field due to less flow resistance share the same pressure profile). Considering annulus containing the infinitesimal, small closed rings (closed boxes) the pressure is balanced in each of these boxes. For this purpose, on top of the axial flow, tangential flow can be observed in the micro scale. This is associated with the circular connected structure of the wellbore, which allows fluid transportation from the narrow side to the wide side. **Figure 4.2** has a better illustration for this explanation.

![Figure 4.2: Physical explanation for balanced pressure at the surface of a cross-section.](image-url)
According to Figure 4.2, in the case that this study concerns, the container (wellbore) is not open-ended, instead it is a closed box bounded by boundary conditions. Therefore, the role of velocity on pressure (as ruled by the governing equation) is absorbed as the boundary conditions dictate. At the random depth h, the flow will try to equalize pressure all along the cross-section. Because, the pressure distribution at the inlet (bottom), and at the outlet (top surface) are given evenly, any point, the solver will try to resemble these conditions. When the enlarged cross-section is viewed, assuming higher velocity is at the wide section; necessarily, higher total pressure is expected in the wide section as well. Analogous to that, lower total pressure is expected in the narrow section. This means that pressure across the same depth must fluctuate. However, for the reason of resembling the flow profile to the boundary conditions, there will be a minor (secondary) flow in the tangential direction to level pressure at the same depth. In other words, in every other infinitesimal section of annulus on the horizontal plane, the flow spirals. Taking the magnitude of flow into account, this tangential flow will be almost negligible.

Note that the discussion above takes only the eccentric annulus with non-rotating inner pipe into the account. If the concentric case is considered, the annulus is expected to have uniformly distributed pressure at the same depth cross-section. For the eccentric annulus with rotating inner pipe however, it is more complicated. In this case, along with axial velocity, tangential velocity is introduced and depending on the inner pipe angular velocity, their dominance change. Also, the boundary conditions are altered in that case. A stationary outer wall and moving inner wall are added to the boundary conditions set. Shear rate distribution, rotational Taylor’s number and Reynolds number all affect the pressure profile of an annulus with a rotating inner pipe.

Abiding to the non-Newtonian fluid characteristics, power law model fluids react to yield stress effects. For this study, this reaction is in the form of shear thinning. The shear thinning effect causes viscosity reduction and that impacts Reynolds number (potentially flow regime), and pressure gradient. For the eccentric geometries discussed in this section, the magnitude of the eccentricity change the shear
rate applied across the geometry and that results in modification of the variation of shear rate dependent 
viscosity in different sections of the flow domain.

All in all, the eccentricity governed geometry (due to clearance (flow diameter) that flow occurs), 
redefined viscosity (µ) and fluid velocity (v) counter acts on Reynolds number. To recall, dimensionless 
Reynolds number equation is:

\[ N_{Re} = \frac{\rho v D}{\mu} \]  

(4.2)

where \( N_{Re} \) is Reynolds number, \( \rho \) is density, \( v \) is fluid velocity, \( D \) is flow diameter and \( \mu \) is 
viscosity.

### 4.1.1 Concentric Annulus With Laminar Flow and Without Rotation

**Figure 4.3** shows the 3-d pressure contour plots for the concentric annulus with a non-rotating 
inner pipe. As mentioned previously, tones of colors in the label bar represent the magnitude of pressure 
profile in the annulus. This is the most basic case, therefore not much irregularity is expected. The 
pressure is gradual distributed from inlet to outlet. When the cross sectional pressure on the inlet surface 
is analyzed, it is seen that pressure distribution is uniform. The pressure clip plot, especially Figure 4.3.c 
shows it clearly.

**Figure 4.4** includes the 3-d velocity contour plots for the same case. The most striking 
observation is on the fluid velocity at the walls and at the center. The figures verify the no slip boundary 
fluid flow with fluid is stationary at the walls and the maximum fluid velocity is reached at the center of 
annular clearance. This figure also shows that a fully developed fluid flow is active; this can be deduced 
from the fact that there is no hydraulic entrance effect observed in the fluid velocity profile. If a line was 
drawn from inlet to outlet, axial distance vs. the velocity data set on that line would show a constant trend. 
Since this case is concentric, there is no wide and narrow side in the annulus. This eliminates the velocity 
variation on the same surface. The parabolic velocity profile is effective on any portion of cross sectional 
area at the same depth.
4.1.2 30% Eccentric Annulus With Laminar Flow and Without Rotation

When the geometry is changed to 30% eccentric annulus, there occurs a minor change in the pressure contour plot. As the label residing next to the contour plots are analyzed, it is seen that the maximum pressure amount is smaller as compared to the previous case. To recall, the maximum pressure was 14.92 psi in the previous case, this time it reduces to 14.895 psi. The difference between the maximum and the minimum values show the pressure loss along the well in the annulus. After all this description, it is possible to state that the pressure drop across the annulus decreases as eccentricity increases. This result was justified in the discussion above with shear stress and viscosity relation. In the general outlook of the pressure distribution, the only factor changing is the amount of pressure drop. The uniform pressure distributions on the surface at any depth and gradual pressure decline from inlet to surface are maintained. Figure 4.5 illustrates these with the 3-d pressure contour plots for the 30% eccentric annulus with non-rotating inner pipe. From the pressure standpoint, the findings are somehow consistent with the concentric annulus case. The pressure distribution is gradual from inlet to outlet. As discussed previously, pressure is the same on an inlet surface and is evenly distributed on the surface of at a random depth. The pressure clip plot shows it clearly.

The obvious change is observed in the velocity graphs. Increments in the eccentricity cause the velocity to be distributed unevenly. In a comparison to a concentric case, it is seen that the velocity is higher at the wider portion of the cross-sectional area and lower in the narrow section. Same as the pressure contour plots, the velocity contour plots show the maximum and the minimum values on the scale next to the contour plots. An uneven profile of velocity is striking at first look. But more importantly, when the quantities in the label scale are compared to the quantities in concentric geometry case, it indicates that the magnitude of the maximum velocity is greater in this case. Figure 4.6 presents the 3-d velocity contour plots. Still, no slip boundary type fluid flow is effective. The fluid velocity at the walls is zero. The maximum fluid velocity is reached at the center of annular clearance. With this case, the wide and narrow sides of the geometry are now populated. There is a necessity to emphasize again
that stress levels in these portions define the flow rate and dependently fluid velocity. This yields velocity variation on the same surface.

4.1.3 60% Eccentric Annulus With Laminar Flow and Without Rotation

As the last step increment of eccentricity, 60% eccentric annulus geometry is investigated. Figure 4.7 presents 3-d pressure contour plots for the 60% eccentric annulus with a fixed inner pipe. The only change is observed on the magnitude of the pressure drop along the annulus. The trends are consistent with the previous cases with uniformly distributed pressure on the cross sectional area and gradually decreasing pressure from velocity inlet to pressure outlet. It can be concluded from these examples that the pressure loss trend (exception to the magnitude) in the annulus does not change much with increasing eccentricity, providing that no rotation is introduced. For the magnitude case, it can be concluded that the eccentricity and the pressure loss amount have inverse proportion. This is mainly related to the shear thinning effect that power law fluid model possesses.

The velocity curves significantly change in this scenario. In the 30 % eccentric geometry case, the velocity diversity between the narrow and the wide side is moderate when compared to this case. As the level of eccentricity increases to 60 %, the fluid velocity in the narrow area gets closer to zero. The color of velocity profile in the wide section and the velocity profile on the stationary walls are almost the same. Also, it can be seen that the peak velocity is higher. The peak velocity has started with 4.92 ft/sec in the concentric annulus, respectively it increased to 7.8 ft/sec and 8.7 ft/sec. Such a jump in the magnitudes shows the level of unevenly shared resistance in the annulus that fluid encounters. The shear rate that the fluid flow will be exposed to varies from narrow section to wide section. All of these alterations in fluid velocity are illustrated in Figure 4.8 with the 3-d velocity contour plots. It can be concluded from the velocity examples that the eccentricity causes uneven velocity distribution on the cross sectional surface area and the increments in eccentricity make the uneven distribution even more severe.
Figure 4.3: 3-d pressure contour plots for concentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus evenly through whole body. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure 4.4: 3-d velocity contour plots for concentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus evenly through whole body.
Figure 4.5: 3-d pressure contour plots for 30% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure 4.6: 3-d velocity contour plots for 30% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure 4.7: 3-d pressure contour plots for 60% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure 4.8: 3-d velocity contour plots for 60% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Another way to observe the effect of eccentricity on velocity is demonstration in an X-Y plot. Parabolic velocity profiles across the annulus in the horizontal plane are shown in detail. In the Figure 4.9 below, the variation of fluid velocity is clearly presented. These curves represent lines passing in a horizontal plane perpendicular to the annulus. The line passes through the narrowest and the widest sections of the annulus in the horizontal plane at the inlet surface. That line resides at the bottom of cylindrical body in Figure 4.1. The left side of the graph stands for the wide side and the right side of the graph stands for narrow side. The blank area in the middle is the casing string. It is important to notice that this graph is drawn for annular velocity. The flow inside the casing is not of interest here. The ends of the individual curves indicate the presence of walls.

![Velocity profile at inlet](image)

Figure 4.9: Velocity distributions across annulus for non-rotating inner pipe cases. Dashed lines indicate the difference between narrow and wide side of the annulus.
Same as the velocity, the pressure profile is attributed with a detailed X-Y plot method. **Figure 4.10** and **Figure 4.11** show the pressure distribution through the imaginary lines residing along an annulus. Those lines were shown in Figure 4.1. Figure 4.10 represents the pressure trend embedded to the line passing next to the wellbore. Figure 4.11 represents the pressure trend embedded to the line passing passing next to inner pipe (casing). The curves for both sides exactly match. This shows that pressure is uniformly distributed on the same surface. The decreasing trend of total pressure drop for increasing eccentricity is clearly shown. In the x-axis, the inlet point is where axial distance equals to 0 and the outlet is at 72 in.

![Pressure drop in the annulus, near wellbore](image)

**Figure 4.10**: Pressure drop in the annulus profiles for non-rotating inner pipe cases. Pressure drop near wellbore.
4.2 Effect of Casing Rotation and Eccentricity

On top of the eccentricity and flow rate distribution justification revealed in the previous section, the rotational motion of inner pipe is introduced in this section. From this point on, the results and interpretations are for the casing string rotated with a rotational speed of 80 rpm, which is an applicable rate for the non-retrievable CwD operations. Note that in the retrievable system, the rotational speed of the drilling bit can be adjusted with a downhole motor. Therefore, the actual angular speed that the casing string has can be regulated.

For the previous stationary walls condition, the velocity profile on the cross-sectional surface showed fluctuations due to an eccentricity influence on flow rate. In this section, the boundary conditions have changed such that the inner pipe is not stationary anymore. Rotation is present and because of this, the inertial effects are involved in the process. So, the inertial effects as a result of geometric irregularities
and rotational act of fluid actively affect the result. Since the inner cylindrical wall is not stationary, the fluid velocities in the tangential and the horizontal directions do not vanish and the mass conservation equation applies in the axial, tangential and horizontal directions.

Just as discussed in the previous section, the shear thinning effect of power law fluids is always present. For the first case of this section, the same situation is valid, because the convective terms disappear due to a lack of geometrical irregularities. However, when eccentricity exists, the force acting on the applied pressure gradient is a combination of spatially-dependent convective term, secondary flows and vortices, and shear thinning. These terms were described in the previous chapters. The combination of these three influences is mainly directed by drilling fluid density, rotational speed and flow rate.

Chin, in his book, described a similar case to the scenario and he concludes that for a fixed pressure gradient, the required flow rate increases as the inner pipe rotates. The author further related this to the decreased apparent viscosity. In addition to the reduced apparent viscosity, it can be deduced from the explanations that the effect of helical flow caused by the rotation distributes the resistance load between axial and tangential flow such that the axial flow shoulders less load. In this study, instead of a fixed pressure gradient, a fixed flow rate is present. Regarding the intrinsic relation described by Chin, the pressure gradient decreases (Chin, 2011). How shear thinning and convective terms counteract is shown in the comparative graph that is available at the end of this chapter.

4.2.1 Concentric Annulus With Laminar Flow and With Rotation (80rpm)

The figures starting from Figure 4.3 to Figure 4.11 consider the scenarios with a stationary inner pipe condition. However, the figures beyond this point show how a constantly rotating pipe can change the profiles of velocity and pressure. To start with, Figure 4.12 shows the 3-d pressure contour plots for the concentric annulus with a rotating inner pipe. In comparison to the non-rotating case, it is seen that the pressure distribution is no longer uniform on the cross sectional area. The pressure values are smaller at the region next to the inner rotating wall. That is related to the centrifugal effect and azimuthal motion of
inner pipe and fluid abiding to this behavior. As the distance (in horizontal plane) from the source of motion adds up, the effect diminishes and the pressure values increase. Despite the difference in quantities on the cross sectional area, gradual pressure drop along the annulus trend is shared. For a clearer presentation, pressure clip plots can be analyzed.

Figure 4.13 includes the 3-d velocity contour plots for the same case. The most striking point about the velocity contour plots is related to velocity trends at the walls. No-slip boundary conditions are not satisfied anymore, since the casing is not stationary. As a result of rotating action of inner pipe, the fluid velocity next to the pipe is nonzero; in its place, the resultant velocity is a value also monitored by the rotational speed. As a result of concentric annulus, the narrow and wide annulus terms are introduced. The maximum fluid velocity still occurs at the center of annular clearance. The velocity is distributed by a parabolic profile on a cross sectional area at any depth. The violation in no slip boundary condition is obvious, however the fully developed fluid flow is preserved in this case. Whatever fluid velocity is observed in the velocity inlet, it is the same in the pressure outlet.

4.2.2 30% Eccentric Annulus With Laminar Flow and With Rotation (80rpm)

Changing the well geometry from concentric to 30% eccentric, creates a reduction in the pressure drop. But the magnitude of that reduction is not the same in comparison to a concentric case. The physical explanation for this fact is given by the coexistence of the forces reducing and fostering the apparent viscosity. To recall, in non-rotating inner pipe cases, only shear thinning effect of power law fluid model acts on the annular pressure loss gradient. In these cases, secondary forces occurring in the annulus and the inertial effect attempt to increase the annular pressure loss as opposed to shear thinning effect. Therefore, a direct conclusion on these types of cases can not be drawn. Figure 4.14 illustrates this with the 3-d pressure contour plots for the 30% eccentric annulus with rotating inner pipe. The figures are somehow consistent with the ones for the concentric annulus. Pressure distribution is gradual from inlet to outlet. As discussed earlier, the pressure is irregularly distributed on the surface at a random depth.
The velocity plots for 30% eccentric annulus clearly exhibit the influence of tangential flow by distributing the peak velocity region along the flow direction. These plots qualitatively make the author to observe the peak velocity smoothened along the direction of tangential movement. Also, in quantities, the maximum fluid velocity is not observed in the widest section of annulus anymore. The tones of color show that the peak velocity has moved towards the narrow area in a reduced fashion. In consistent with the previous case, the fluid velocity is nonzero at the inner wall due to fluid abiding to rotating inner pipe. Figure 4.15 presents the 3-d velocity contour plots.

4.2.3 60% Eccentric Annulus With Laminar Flow and With Rotation (80rpm)

In this case, the trends observed in the previous inner pipe rotating cases are valid for the pressure perspective. Only the level of frictional pressure loss is different depending on eccentricity and the factors linked to it; that is explained previously. Figure 4.16 presents 3-d pressure contour plots for the 60% eccentric annulus with a rotating inner pipe. It can be summarized that the combination of constant inner pipe rotational motion and eccentricity has changed the pressure distribution on the cross sectional area such that the pressure adjacent to stationary outer wall is greater than the pressure adjacent to rotating inner pipe. Despite this new trend, increasing eccentricity has failed to change global gradual pressure loss trend along the annulus.

All of the fluid velocity alterations are illustrated Figure 4.17 with the 3-d velocity contour plots. Distributing influence of tangential flow is observed in a consistency with the previous case. Fluid velocity is at a minimum at the stationary wall. It can be concluded that combination of constant inner pipe rotation and eccentricity changes the fluid velocity profile significantly. While eccentricity causes the uneven fluid distribution on the cross sectional surface, the helical flow due to inner pipe rotation helps to weaken the magnitude of peak velocity and share it along the annulus. To recall, for 60% eccentric geometries, it was 8.7 ft/sec in the non-rotating inner pipe scenario, here in the rotating case it is 7.5 ft/sec.
Figure 4.12: 3-d pressure contour plots for concentric annulus and with inner pipe rotation @ 80 rpm. Flow is in $+y$ direction. (a) Pressure contour plot in $Y-Z$ plane. (b) Pressure contour clip plot. Clip plane cuts annulus evenly through whole body. (c) Pressure contour plot in $X-Y-Z$ plane showing cross-section of inlet.
Figure 4.13: 3-d velocity contour plots for concentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus evenly through whole body.
Figure 4.14: 3-d pressure contour plots for 30% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus evenly through whole body. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure 4.15: 3-d velocity contour plots for 30% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus evenly through whole body.
Figure 4.16: 3-d pressure contour plots for 60% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus evenly through whole body. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure 4.17: 3-d velocity contour plots for 60% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus evenly through whole body.
X-Y plot in Figure 4.18 verifies what has been explained for the rotating inner pipe cases. Although the velocity magnitude distribution influence across the surface is not as clear as it is demonstrated in the contour plots, magnitudes and trends in the narrowest and the widest sections are available. As can be seen, in the narrow section, the parabolic trend is not smoothly applied. These curves represent a line passing in horizontal plane perpendicular to the annulus. The line passes through the narrowest and widest sections of annulus in the horizontal plane at the inlet surface. That line resides at the bottom of cylindrical body in Figure 4.1. The left side of the graph stands for the wide side of the annulus and the right side of the graph stands for the narrow side of the annulus. The blank area in the middle is the casing string. The data generated at the vicinity of the inner pipe is not necessarily well simulated. The distortion is related to boundary condition effect. To recall, boundary conditions for the rotating inner pipe scenarios include angular inner pipe velocity of 80 rpm in addition to stationary outer wall, inlet velocity and outlet flow.

Figure 4.18: Velocity distributions across annulus for rotating inner pipe cases.
Likewise, the pressure profile is elucidated with a detailed X-Y plot method. Figure 4.19 and Figure 4.20 show the pressure distribution through imaginary lines residing along the annulus. Those lines reside along the annulus of a cylindrical body were shown in Figure 4.1. Figure 4.19 represents the line passing next to the wellbore. Figure 4.20 represents the line passing next to the inner pipe (casing). The fact that pressure is not distributed evenly on the surface of a cross-sectional area is verified with these figures. The magnitudes of pressure loss are diverse near the wellbore and near the casing. Not only the level of pressure loss, but also the trend near the outlet is different. Both curves meet in the designated outlet pressure boundary. With the contribution of the vortices and tangential movement, and centrifugal effect near casing pressure drop values are always smaller than the ones by near wellbore pressure. As fluid goes away from the source of the rotation in horizontal plane, impact of the helical flow vanishes. A decreasing trend of total pressure drop for increasing eccentricity is clearly shown. In the x-axis, an inlet point is where axial distance equals to 0 and at outlet it is 72 in. Due to the boundary conditions of velocity inlet and pressure outlet, the data simulated at the entrance and the data at the outlet are distorted. For that reason these figures show the well-modeled interval. The essence of these two figures is to show the pressure drop trend.

Figure 4.19: Pressure drop in the annulus profiles for rotating inner pipe cases. Pressure drop near wellbore.
Finally, as marked earlier, increasing eccentricity decreases the frictional pressure loss. This has been linked to shear thinning effect by drilling fluid that has less resistance because shear thinning reduces the effective fluid viscosity, and convective terms that result from the eccentricity induced geometric irregularities and inertial forces. Figure 4.21 presents the final comparison of frictional pressure losses for the sixteen different cases run in this study.

Figure 4.21 presents the frictional pressure loss in two terms: pressure loss gradient (psi/ft) and equivalent mud weight (EMW) (ppg). The main purpose on doing so is to emphasize the magnitude of the pressure loss along annulus. Regarding the accuracy of the model and limitations on the model construction and computational effort, this model is run in geometry of 6 ft. interval. Rather than looking for the whole well geometry, the main purpose sought is to describe pressure loss along the annulus in gradient form or EMW form. For the practical purpose, in order these results to make sense, converting these findings to real well depths by multiplying with true vertical depth and extrapolating to global scale of the wellbore (whole well environment starting from the bottomhole to surface) can help.
Figure 4.21: Comparison of pressure drop in the annulus profiles for stationary inner pipe and rotating inner pipe cases.
CHAPTER 5

CONCLUSIONS, DISCUSSION AND FUTURE WORK

In this study, Casing while Drilling (CwD) technology and its practical aspects are analyzed. This task was accomplished by creating a 3-d CFD model and simulating the conditions as it is in the CwD operations. For the sake of research, the effect of eccentricity and the presence of continuous uniform rotational motion of an inner pipe on mud displacement processes in the specific CwD geometries were investigated from the point view of frictional pressure loss and fluid flow velocity. The simulation results verified by the numerical solutions shed light onto some unclear points about hydraulics of narrow annuli in CwD.

The following conclusions can be drawn upon findings from this study:

- Eccentricity is an important parameter on annular hydraulics. As an alone parameter, eccentricity has an inverse proportional relation with annular frictional pressure loss. As the casing string eccentricity is increased, the pressure drop is decreased. This is bound to the shear rate dependency of the power law fluid viscosity. This behavior summarizes the axial trend in a vertical well. However, the results also showed that eccentricity by itself is not enough to disturb uniform pressure distribution in a cross-sectional surface at any depth. In other words, pressure distribution at the narrow side of the well matches with the one in wide side. Yet, the uniformity is not preserved in the fluid velocity profile. The eccentric geometry leads the fluid velocity to be high in the wide side and to be low in the narrow side of the annulus.

- The combination of rotation and eccentricity failed to yield a direct conclusion. However, it can be stated that shear rate intrusion on drilling fluid viscosity is resisted by the inertial forces to finalize the frictional pressure loss. This results in a more moderate path in the frictional pressure loss comparison along the annulus. It can also be concluded that this
combination partially alters pressure distribution on same cross-sectional surface at any depth. The distortion does not happen in conjunction with tininess in the annular clearance as in the velocity profile, but it happens as a result of rotation versus stationary condition. Adjacent regions to the dynamic pipe yield a lower pressure value as compared to the regions adjacent to the stationary wall. The rotation also works to distribute the peak of fluid velocity caused by eccentricity induced geometric irregularity.

- A final conclusion may be addressed to successful results of the CFD method. It is seen that CFD was capable of simulating the conditions realistically.

Presenting the pressure distribution and commenting on it makes this study one of a kind. Considering that ECD contribution to unique plastering effect of CwD is still being researched, the pressure description provided here can constitute a supplementary source for that kind of research incentive.

5.1 Discussion and Future Work

It is possible to extend this research in various ways. In this study, a single data set was used. Alternatively, the input parameters can be altered to obtain results for new cases. In the long run, the effect of every parameter can be investigated by having a control group and a variable group. With that approach, it is possible to interpret the significance of every parameter by the magnitude and the way of their impact. For the verification of results, stationary inner-pipe cases are solved numerically. It is quite impossible to find analytical solutions for every single case. Considering that, the only option available was to comment on the results backing up with physical theories. An experimental study or field data would be handy to conclude a compact validation. This study avoids using a turbulent flow regime due to its erosive characteristic. Rather than supporting the plastering effect, this would prevent the plastering processes. Also, it would cause variation in wellbore diameter, fluctuating casing stand-off, and centralization.
The plastering effect of CwD is a comprehensive topic to research. Drilling fluid characteristics, formation characteristics and hydraulics are some components of this complicated problem. In order to be able to name the plastering effect as a function of ECD, all these aspects must be worked either experimentally, analytically or with field data. These works can be related to the geomechanical structures of the rocks of interest, microfracture generation investigation, drilling fluid additives contribution, chemical and physical interactions while filter cake is formed, and drilling fluid invasion processes. Although the link between annular pressure distribution and the plastering effect is only proven in qualitative manner empirically, this study will help to visualize ECD for the CwD annuli. As the contribution of ECD to the plastering effect of CwD technology is proven, this study will present a background work.

Regarding the model inputs and scenarios simulated, this study considered laminar flow regime since in vertical wells, exceeding the cutting setting velocity in magnitude is enough to transport cuttings. However, expanding this study to horizontal wells will require a turbulent flow regime to effectively clean the wellbore. In that scenario, the trends would change and ECD would be more critical. This study can also be extended to the gasified drilling fluid modeling to simulate conditions in underbalanced CwD operations. In this case, in addition to the pressure term, the contribution of temperature in annular flow field will be active. The governing equations will include compressibility of drilling fluid with two phase fluid flows.
REFERENCES


APPENDIX A.

FIGURES SHOWING EFFECT OF ECCENTRICITY AND INNER PIPE ROTATION ON ANNULAR PRESSURE AND VELOCITY PROFILES
Figure A.1: 3-d pressure contour plots for 10% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.2: 3-d velocity contour plots for 10% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.3: 3-d pressure contour plots for 20% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.4: 3-d velocity contour plots for 20% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.5: 3-d pressure contour plots for 40% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.6: 3-d velocity contour plots for 40% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.7: 3-d pressure contour plots for 50% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.8: 3-d velocity contour plots for 50% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.9: 3-d pressure contour plots for 80% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.10: 3-d velocity contour plots for 80% eccentric annulus and without inner pipe rotation. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.11: 3-d pressure contour plots for 10% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.12: 3-d velocity contour plots for 10% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.13: 3-d pressure contour plots for 20% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.14: 3-d velocity contour plots for 20% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.15: 3-d pressure contour plots for 40% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.16: 3-d velocity contour plots for 40% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.17: 3-d pressure contour plots for 50% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.18: 3-d velocity contour plots for 50% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.19: 3-d pressure contour plots for 80% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Pressure contour plot in Y-Z plane. (b) Pressure contour clip plot. Clip plane cuts annulus through the widest and narrowest sections. (c) Pressure contour plot in X-Y-Z plane showing cross-section of inlet.
Figure A.20: 3-d velocity contour plots for 80% eccentric annulus and with inner pipe rotation @ 80 rpm. Flow is in +y direction. (a) Velocity contour plot in X-Y-Z plane. (b) Velocity profile at cross sectional profile of annulus at inlet surface. (c) Velocity contour clip plot. Clip plane cuts annulus through the widest and narrowest sections.
Figure A.21: Velocity distributions across annulus for all non-rotating inner pipe cases.
Figure A.22: Pressure drop in the annulus profiles for all non-rotating inner pipe cases. (a) Pressure drop near wellbore. (b) Pressure drop near casing.
Figure A.23: Velocity distributions across annulus for all rotating inner pipe cases.
Figure A.24: Pressure drop in the annulus profiles for all rotating inner pipe cases. (a) Pressure drop near wellbore. (b) Pressure drop near casing.