World Academy of Science, Engineering and Technology
International Journal of Geotechnical and Geological Engineering
Vol:12, No:10, 2018

An Automated Approach to the Nozzle Configuration of Polycrystalline Diamond Compact Drill Bits for Effective Cuttings Removal

R. Suresh, Pavan Kumar Nimmagadda, Ming Zo Tan, Shane Hart, Sharp Ugwuocha

***Abstract*—**Polycrystalline diamond compact (PDC) drill bits are extensively used in the oil and gas industry as well as the mining industry. Industry engineers continually improve upon PDC drill bit designs and hydraulic conditions. Optimized injection nozzles play a key role in improving the drilling performance and efficiency of these ever changing PDC drill bits. In the first part of this study, computational fluid dynamics (CFD) modelling is performed to investigate the hydrodynamic characteristics of drilling fluid flow around the PDC drill bit. An Open-source CFD software – OpenFOAM simulates the flow around the drill bit, based on the field input data. A specifically developed console application integrates the entire CFD process including, domain extraction, meshing, and solving governing equations and post-processing. The results from the OpenFOAM solver are then compared with that of the ANSYS Fluent software. The data from both software programs agree. The second part of the paper describes the parametric study of the PDC drill bit nozzle to determine the effect of parameters such as number of nozzles, nozzle velocity, nozzle radial position and orientations on the flow field characteristics and bit washing patterns. After analyzing a series of nozzle configurations, the best configuration is identified and recommendations are made for modifying the PDC bit design.

***Keywords*—**ANSYS Fluent, computational fluid dynamics, nozzle configuration, OpenFOAM, PDC dill bit.

I. Introduction

PDC drill bit technology is most widely used for oil drilling tools in recent years. Drill bit hydraulics have been recognized as the major factor influencing the drilling performance [1]. The purpose of proper hydraulic design of drill bits is to have appropriate conditions of drilling fluid flow rate, cooling of PDC cutting structure and also required bit pressure drop to meet Measurement While Drilling tool telemetry requirements [2].

Poor hydraulic design causes improper bottom hole cleaning, which may result in balling (the accumulation of cuttings on the bit face) that decreases the ROP, or may halt drilling in severe cases [3]. To facilitate the improvement in drilling hydraulics, the fluid flow pattern should be optimized in terms of the pressure distribution and the velocity profile in around the PDC Cutters and also in the open area in between blades referred to as the “face volume”. With the help of computational methods, the influence of bit design factors can be understood and utilized for improving the bit hydraulics. An effective approach to improve flow patterns is to optimize the position and orientation of the injection jet nozzles on the drill bit. CFD is one of the computational tools, efficient to simulate the fluid flow phenomena in complex geometry of drill bit design. The use of CFD in optimizing drill bit design will provide inexpensive and reliable results to improve the drilling performance [2].

Suresh R is with the Mechanical Engineering Division, Weatherford Engineered System Support, Mumbai, India (phone: +91 22-66059506; e-mail: ramarao.suresh@weatherford.com).

Pavan Kumar Nimmagadda is with the Software Division, Weatherford Engineered System Support, Mumbai, India (phone: +91 22-66059500; e-mail: pavakumar.nimmagadda@weatherford.com).

II. Drill Bit Hydraulics

Bit hydraulics play an important role in effective cuttings removal away from PDC bit. The study of flow distribution in the bottom hole and on the bit surface is crucial in understanding the cleaning mechanism of the cuttings, hence helps in improving the design of drill bit. The overall hydraulic energy distribution around the bit should be optimized in order to improve the cleaning and cooling efficiencies of fluid flow [4]. The fluid emerging from drill bit nozzle (shown in Fig. 1) should reach at borehole bottom in the optimal angle and should evacuate the face volume in the most direct path to perform an efficient cleaning action. In the meantime the excessive flow velocity and jet impingement on the blade should be avoided eroding blade and brazing materials. Back flow of Eddie current also need prevented to avoid accumulation of stagnated cuttings. Regrinding of these cutters accelerates PDC wear. CFD is a powerful tool to study drilling fluid flow in the bottom hole in a comprehensive way. CFD utilizes computational algorithms to solve partial differential equations pertaining to the flow of fluid. The whole flow domain is resolved into smaller control volumes called finite volumes. Using advanced high speed computers, 3D incompressible Navier-Stokes equations are solved using numerical techniques with specified boundary conditions at each boundary. A converged CFD solution gives the values of flow variables at each control volume, which enables the user to examine the flow field in a comprehensive way within the flow domain. A complete three dimensional flow field around the bit along with recirculation zones, dead zones and areas of high or low momentum flows can be clearly captured by plotting the velocity field at user defined planes with in the flow domain. Designer will thus be able to identify the wetted regions over the bit, flow direction on the annulus of the bit which will help improving in the re-design of the bit.

International Science Index, Geotechnical and Geological Engineering Vol:12, No:10, 2018 waset.org/Publication/10009680

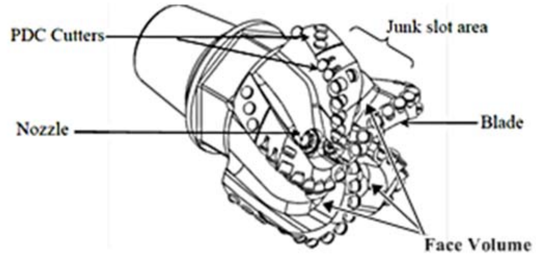


Fig. 1 PDC drill bit

III. CFD Modeling

The present work uses CFD simulations to assess the performance of PDC drill bit for various configurations of nozzle for effective cuttings removal. 3D CAD models of the flow domain are generated using CreO software and open-source tools are used for the CFD process steps. An open-source tetrahedral mesher called GMsh is used for meshing purpose [6], OpenFOAM is used as CFD Solver [7] and ParaView is used as post-processing tool [8]. Further details of each step are given in later sections.

A. Fluid Model

In the present study, water is considered as the drilling fluid which is treated as Newtonian and incompressible fluid. Non-Newtonian fluids and Multiphase can be simulated with this algorithm. A finite volume model which solves non-linear three dimensional Navier-Stokes equations coupled with appropriate turbulence model is used to resolve the flow field around the drill bit. Both steady and unsteady flow solutions are obtained by using in-built finite volume incompressible pressure coupling and solver schemes that are available in OpenFOAM. Heat transfer or thermal effects and drill bit rotation are not included in the scope of work. The conservation of mass and momentum only are solved for multiple nozzle configurations in OpenFOAM.

B. Governing Equations

The formulation of the model is given below [5]. The conservation of mass equation states that the change of mass inside the control volume is equal to the balance of fluid mass entering and leaving the control volume. The conservation principle is represented through the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho U) = 0 \tag{1}$$

where, ρ is the density, U is the velocity, and t is the time. The first term is the unsteady term which represents the rate of change of density and the second term is the convective term which represents the net rate of mass flow through the control volume.

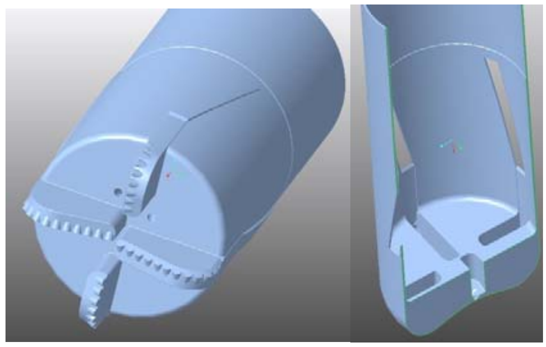
The governing equation for the conservation of linear momentum, written in conservative form is

$$\frac{\partial}{\partial t}(\rho U) + \nabla.(\rho \mathrm{UU}) = -\nabla \mathrm{p} + \nabla.(\tau) + \rho \mathrm{g} + \mathrm{F} \tag{2}$$

where, p is the static pressure, τ is the stress tensor, and ρg and F are the gravitational body force and external forces respectively. In addition to Navier-Stokes equations, two more transport equations for turbulent kinetic energy (k) and specific dissipation rate (ω) are used to represent the turbulence of the fluid which is defined by standard k- ω model [9]. The k- ω model has the advantages of accuracy and robustness near wall region and under adverse pressure gradients and hence can be used to compute the jet impingement flows and highly separated flow more accurately.

C. Computational Geometry

A 300 mm diameter drill bit with four blades and 20 PDC cutters on each blade has been used for the simulation. The validation case consists of 4 nozzles with 16 mm diameter each as shown in Fig. 2 (a). Flow coming out from each nozzle is modeled as inclined jet with ±15 degrees inclination with respect to X/Y axis respectively.



(a) (b)

Fig. 2 (a) PDC Drill bit model (b) Flow domain - Cut Plane view

A flow domain is created in CreO by defining a bottom hole surface and a bore wall surface with a diameter of 308 mm. Thus an annulus region is created which is used to define the flow volume. The cut section of the flow volume is shown in Fig. 2 (b). Meshing the flow domain is complex due to intricate gaps between cutters and the wellbore (gaps are in the range of 0.25 mm). The annulus region is also very small compared to the drill bit diameter (1/100$^{\text{th}}$ dia.). A tetrahedral mesher is a more appropriate choice owing to the complexity of the model. GMsh is a robust open-source mesher which generates good quality tetrahedral meshes over complex domains. It provides several options to the user to control the mesh size locally. It also has the advantage of scripting based mesh generation through which meshing can be accomplished in batch mode or command mode with zero user intervention. So, GMsh is selected as meshing tool. Tetrahedral meshes are created inside the drill bit annulus with local refinements near the inlets, blades and cutter regions. Typical mesh size is around 1 million cells. The mesh section view is shown in Fig. 3.

D. Case Setup

OpenFOAM is used to setup and simulate the CFD problem. Mesh is created using GMsh and imported into

OpenFOAM. A steady, in-compressible flow solver is executed with the following settings,

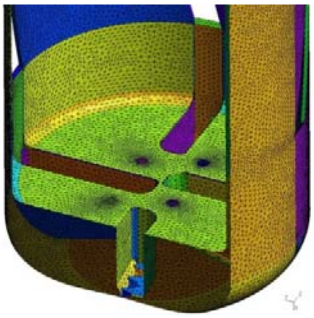


Fig. 3 Tetrahedral Mesh - Cut plane view

1) Boundary Conditions

Drilling fluid enters the domain through nozzles with a specified velocity of 85 m/s. A velocity inlet boundary condition is specified at nozzle surfaces. High speed jets impinge on the drill bore bottom surface and flows through the annulus. A static pressure of 1 atmosphere is specified at the outlet boundary. Both drill bit wall and bore annulus wall are specified as no slip walls.

2) Turbulence

The standard k- ω SST model is used to model turbulence. *Intensity and Mixing Length* option is chosen for inlet turbulence specification. A Turbulence Intensity of 5% and mixing length of 0.1 are used.

3) Solver Settings

Semi-implicit method pressure-linked equations (SIMPLE) algorithm is used for pressure velocity coupling of incompressible flows. Relaxation factors are in the range of 0.3 to 0.7 for pressure, velocity and turbulence quantities in order to have smooth convergence.

E. Results

Converged results from OpenFOAM are loaded in an open-source post processing tool called ParaView. ParaView provides many options to plot the flow variables and visualize the flow field at user specified locations. Fig. 4 (b) shows the velocity contours on a cut plane representing water jets coming out of the nozzle, impinging on the well bore bottom wall and rising up through the annulus. These dead-zones can be the potential accumulation regions of drill cuttings and hence to be avoided. Figs. 5 (a) & (b) show the velocity stream lines along the drill bit annulus for cases without and with guide vanes respectively. Results show that the presence of guide vanes clearly helps redirecting the drilling fluid to all areas of annulus thereby improving the cutting removal mechanism.

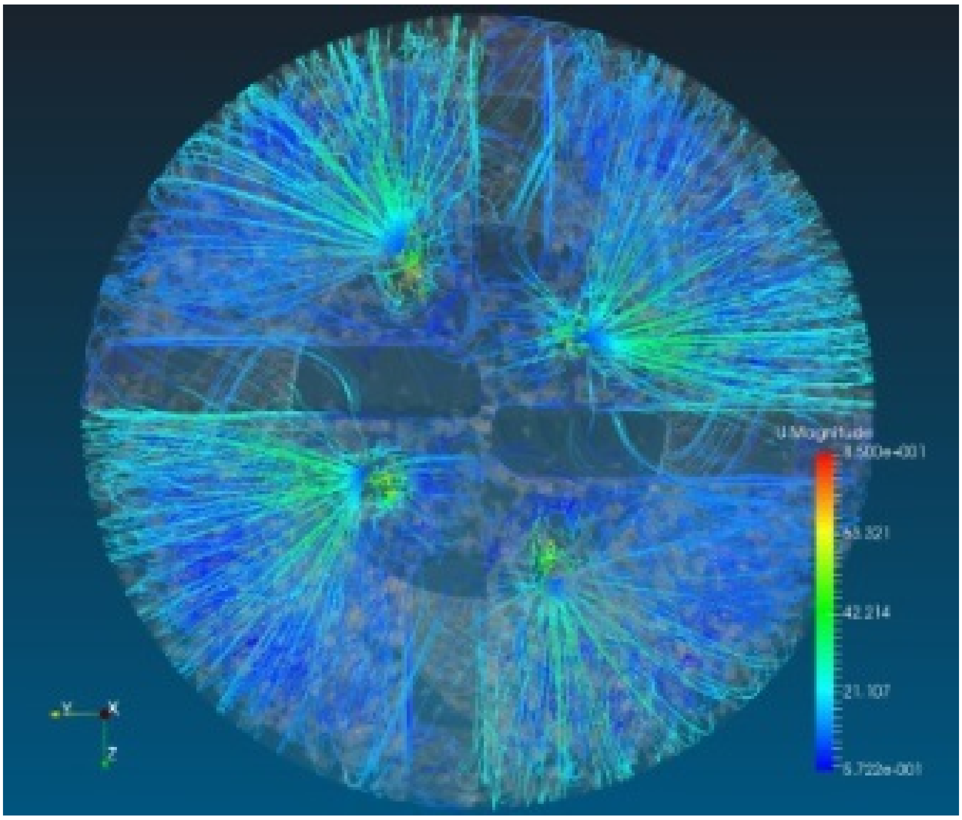


Fig. 4 (a) Streamlines (Bottom view)

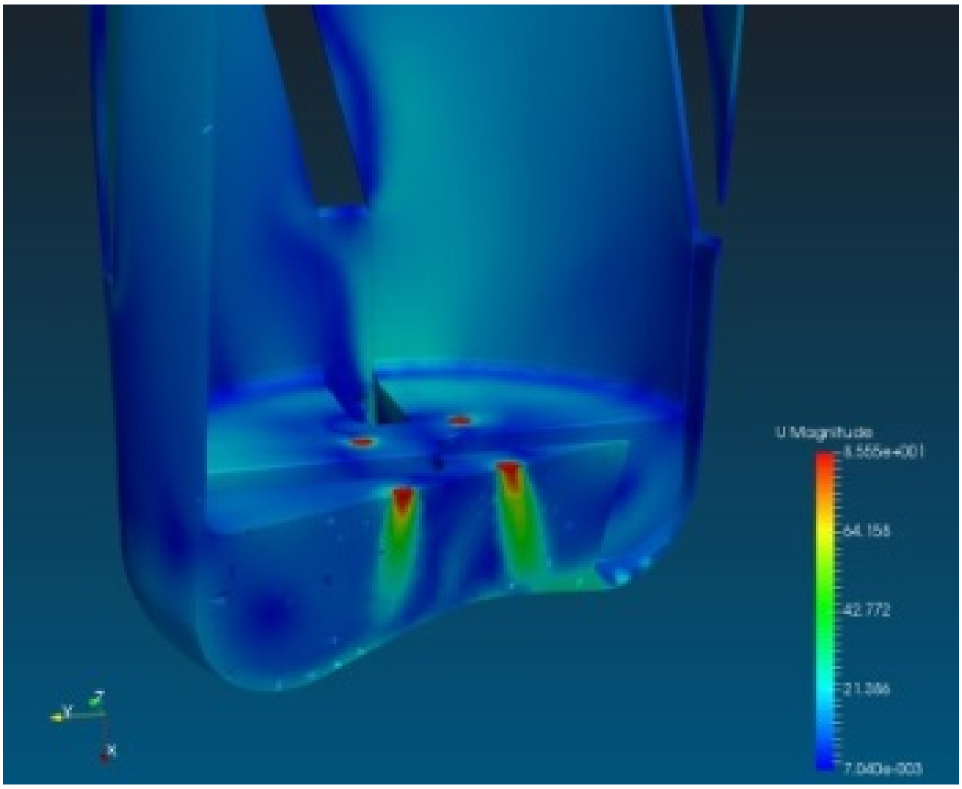


Fig. 4 (b) Velocity contours (cut section)

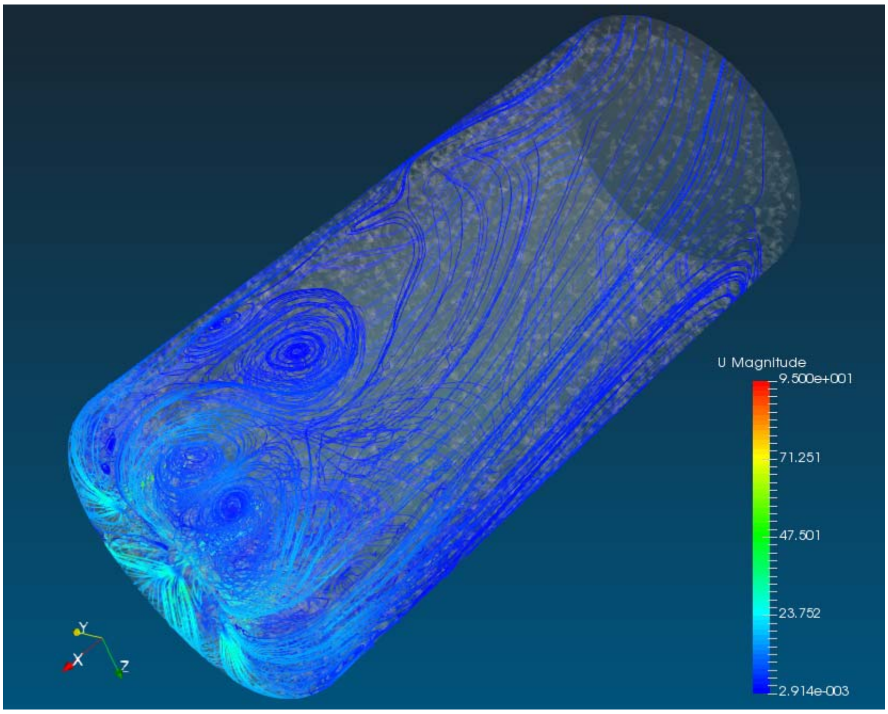


Fig. 5 (a) streamlines -Vaneless annulus

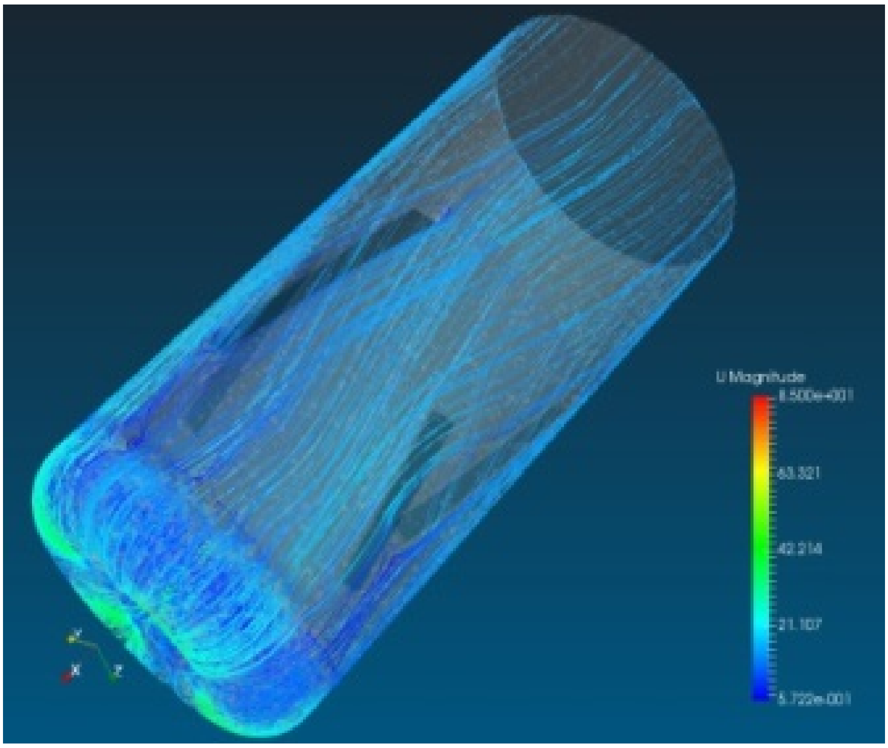
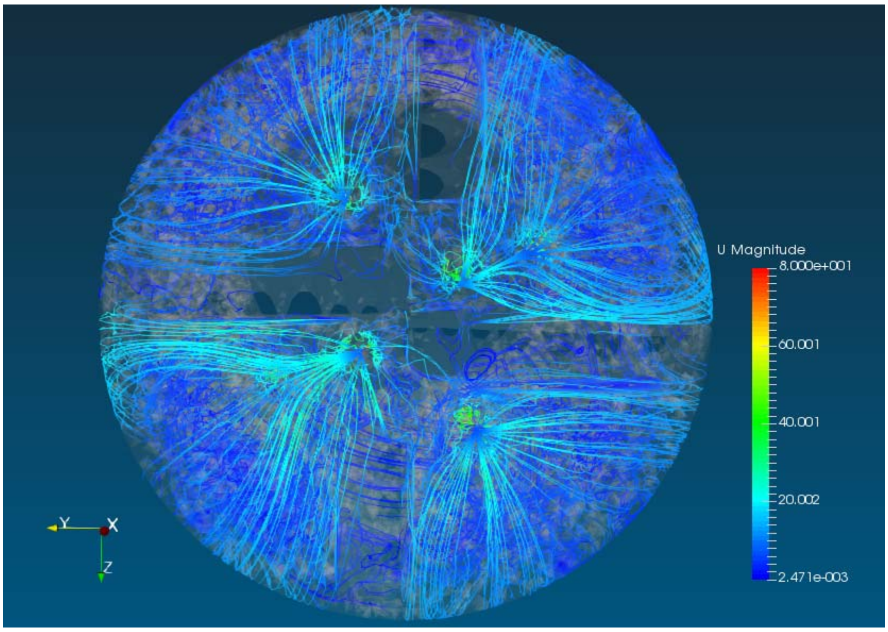


Fig. 5 (b) streamlines - Vaned annulus

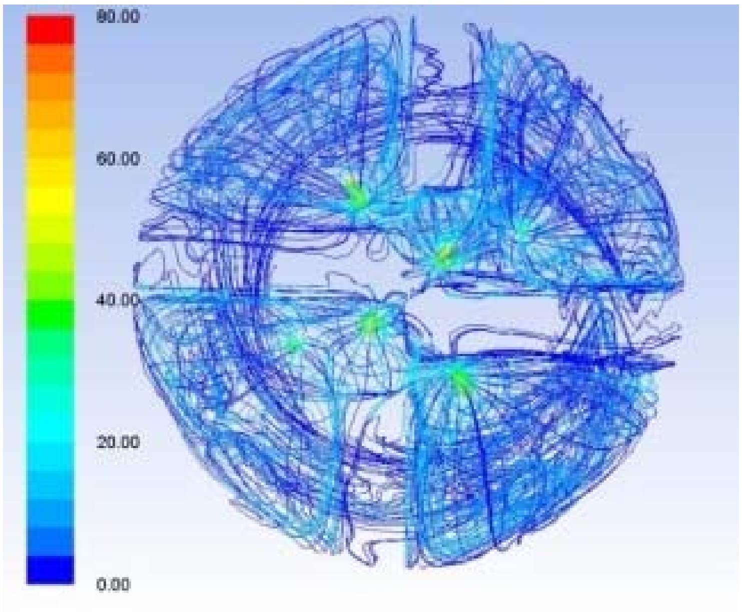
IV. Validation

The OpenFOAM CFD model is validated against the ANSYS Fluent solver [10]. Same mesh that is created in GMsh (as shown Fig. 3) is loaded in both solvers. An inter-code comparison is made by keeping the same boundary conditions, solver settings and convergence criteria.

Fig. 6 (a) shows flow stream lines generated by OpenFOAM from bottom up view and Fig. 6 (b) shows flow streamlines generated by ANSYS Fluent. The stream line patterns match quite well with each other and the stagnation zones appearing at the same circumferential locations. Both software predict the almost the same magnitude of velocity in the bore bottom region.



(a) OpenFOAM



(b) ANSYS Fluent

Fig. 6 Streamline comparisons on well bore (bottom up view)

The velocity distribution given by the two solvers are also compared. Fig. 7 shows the velocity magnitude plotted along a post-processing line, Line-1, running through the well bore diameter below the nozzle. It is found that the results for OpenFOAM and ANSYS Fluent are in agreement.

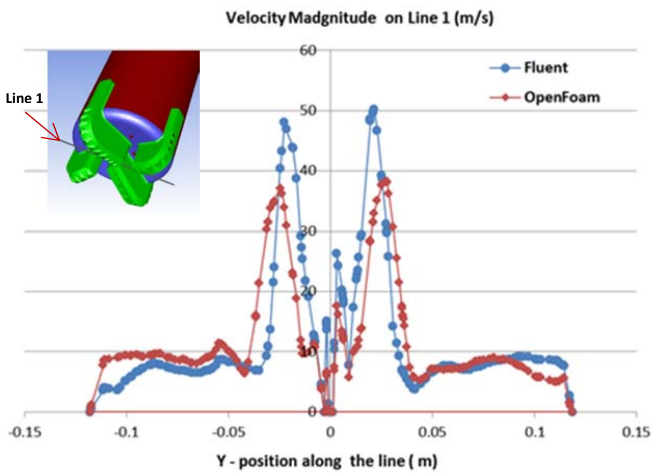


Fig. 7 Velocity magnitude comparisons on post-processing line OpenFOAM vs. ANSYS Fluent

V. CFD Process Automation

At present, there are considerable manual efforts and tool expertise required in executing each of the process steps especially in meshing, setting boundary conditions and post-processing. It is always advantageous for a designer to have an automated process which can help in three ways,

1. Reduced manual efforts on each process step
2. Minimum expertise on CFD tools
3. Faster prototyping of the designs

CFD process automation is accomplished by linking all the processes of the simulation. Individual steps like meshing in GMsh, case set-up in OpenFOAM and post processing in ParaView are run in batch mode by their respective script files. A top level python code is used to link all the processes.

A. Meshing Automation

GMsh is scriptable so that all input data can be parameterized. The scripting language of GMsh allows parameterizing all geometric entities, selecting among various meshing algorithms and options to interface to external solvers etc.

B. Case Setup and Solver Automation

OpenFOAM uses pre-designated folder structure and fixed naming for each of the boundary conditions and solver settings files. OpenFOAM CFD libraries read the input files and solve the governing equations as specified by the user. All the files of OpenFOAM including the mesh files and case setup files are in the text format. They can be re-used for multiple cases and are also editable manually or by any text parser. This feature is particularly useful in modifying any input data (such as changing the mesh or changing inlet velocity) and re-running another CFD simulation without much effort. Multiple cases can be run in series by using any standard scripting languages like Perl/Python which can parse the input file and change a particular portion of it.

C. Post-Processing Automation

An open-source post-processing tool called ParaView is used, which provides several options to automatically extract the plots, contours, vectors of the flow variables at user specified locations and write out in the picture format files.

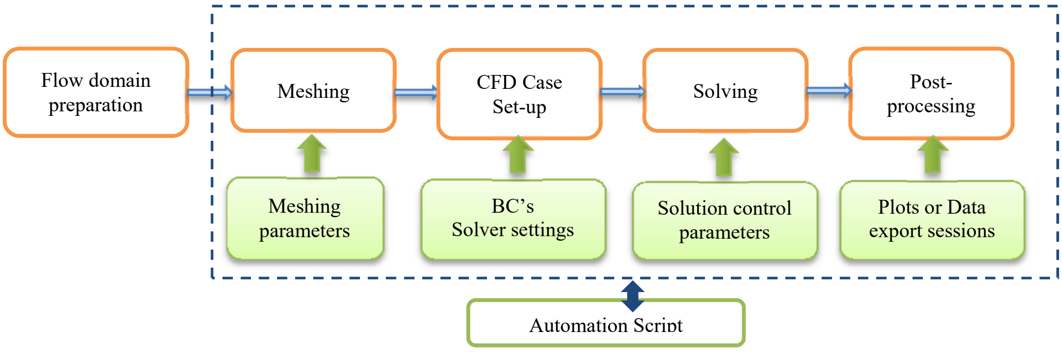


Fig. 8 CFD simulation process flow chart

D.Console Application

A console application is developed with the aim of executing all the above mentioned CFD steps in batch mode with minimal user intervention. A python code is executed in the background to execute one process step at one time or all steps at a time.

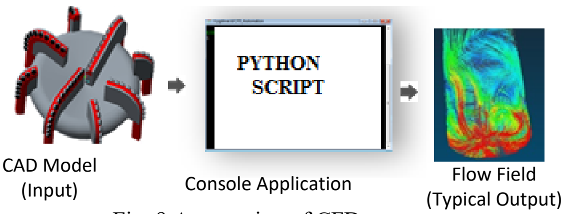


Fig. 9 Automation of CFD process

VI. Parametric Study

In this study, three different nozzle configurations are analyzed so as to understand its effect on drill bit stream-line patterns. A standard four blade and four nozzles PDC drill bit geometry is used for the analysis. Two key parameters identified for the present study, namely nozzle radial position and nozzle tilt angle area areas shown in Table I.

TABLE I
Drill Bit Nozzle Configurations

Nozzle configuration	Nozzle radial positions (mm)	Nozzle tilt angle (degrees from X/Y axis)
Configuration-1	43-43-65-65	0
Configuration-2	60-60-75-75	0
Configuration-3	43-43-65-65	±15.0

VII. Results and Discussion

A series of CFD simulations are run using OpenFOAM for the above mentioned parametric nozzle configurations and results are compared. Post-processing surfaces are created in ParaView, as shown in Fig. 10, for plotting velocity contours and velocity vectors in the well bore region. Velocity contours on the left plane are clipped to the range of 0-10m/s so as to clearly distinguish the high and low velocity flow regions

Figs. 11 (a)-(c) show the velocity contours for three different nozzle configurations. Configuration-1 is a base case. For configuration-2, nozzles are located far from the axis compared to Configuration-1. Configuration-3 is the case of inclined jets with a jet angle of ±15 degrees.

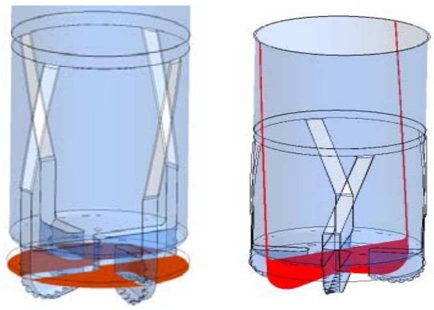


Fig. 10 Post processing planes

During the drilling process, the cuttings are usually directed towards the vertical side walls of the well bore due to the centrifugal force and hydraulic power of the drilling fluid acting on them. For effective evacuation of cuttings, it is desirable to have a maximum flow momentum along the shortest path between the nozzle impingement point to the junk slot area while keeping the recirculation zones to minimum sizes [4], [11], [12].

Comparing Figs. 11 (a) & (b), it is clearly understood that by positioning the nozzles away from the bit axis there is an improvement in the flow velocities in the bit shoulder and gage regions. For Configuration-2, higher velocities (red region) are observed in the well bore regions away from the axis and also fluid enters the annulus region at a higher velocity as compared to Configuration-1. The dead zone (dark blue) for Configuration-2 is smaller in size compared to Configuration-1. Positioning the nozzles away from the bit axis helps improving the flow patterns near the bit shoulder area. However, there is a threshold nozzle position beyond which dead zone or low velocity fluid zones are created near to the bit axis. Also, by comparing Figs. 11 (b) & (c), it is understood that the momentum of the fluid is considerably improved at the shoulder area by providing an inclination to the nozzle jet. Also, dead zone sizes are considerably reduced and it helps to direct more fluid towards the gage area which results in improved cutting removal. So, a combination of radial positioning and jet inclination are to be adopted for the best performance of the drill bit.

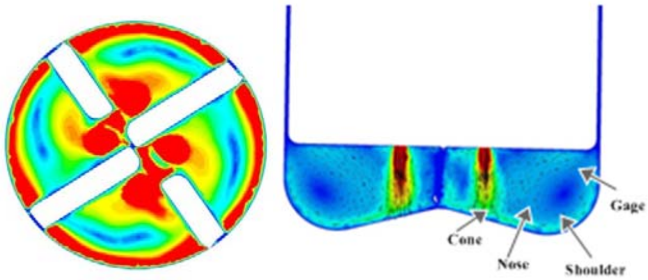


Fig. 11 (a) Velocity Contours - Configuration-1

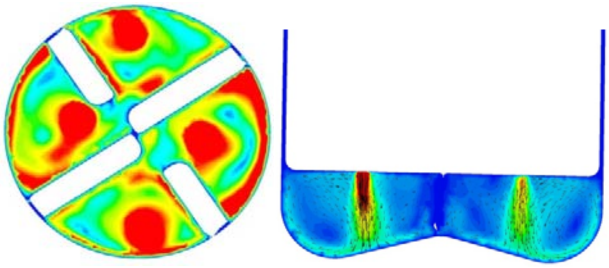


Fig. 11 (b) Velocity Contours - Configuration-2 (Nozzles placed at a larger radius)

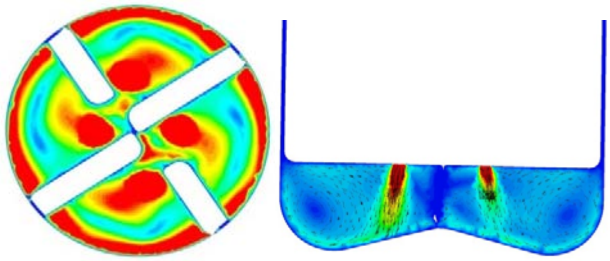


Fig. 11 (c) Velocity Contours - Configuration-3 (Nozzles with ±15 degree inclination)

VIII. Conclusion

In this paper, a three-dimensional model of PDC drill bit is created using CreO software. Flow field around drill bit is analyzed using OpenFOAM. The jet impingement phenomena from the drill bit nozzles and flow behavior inside the well bore region is demonstrated. OpenFOAM results are validated against ANSYS Fluent results.

Process automation of CFD simulation is accomplished by developing a console application. Automation helps the designer to quickly prototype the bit designs and analyzes the flow even without having much expertise on the individual CFD tools.

Finally, a parametric study of three different nozzle configurations is made to know the effect of nozzle position and orientations on the flow behavior. Results showed that an optimum combination of radial positioning and inclination of the nozzle jets improves the cleaning ability of the drill bit.

References

[1] Akin J. et al. "New nozzle hydraulics increase ROP for PDC and rock bits" SPE/IADC drilling conference, pp69-78.

[2] S. Mohan, “PDC Drill bit re-design and simulation for optimized performance, Master of Science Thesis, University of Calgary, Alberta (2014).

[3] Well, M., Marvel, T,. Beuershausen, C “Bit Bailing Mitigation in PDC Bit Design” SPE/IADC Asia Pacific Drilling Technology. 2008.

[4] Zhang H. “Research on the Bottom Hole Flow Field of PDC Bit”, Advances in Petroleum Exploration and Development, Vol. 10, No. 2, 2015, pp. 103-107.

[5] Pletcher, R.H., Tannehill, J.C., and Anderson, D.A. 2013. Computational Fluid Mechanics and Heat Transfer, third edition, 3-9, 66-82. Boca Raton, Florida: CRC Press.

[6] Gmsh: a three-dimensional Finite element mesh generator with built-in pre- and post-processing facilities Int. J. Numer. Meth. Engng 2009; 0:1-24

[7] OpenFOAM: The Open Source CFD Toolbox, User Guide V1706, 2017.

[8] ParaView User guide: https://www.paraview.org/paraview-guide.

[9] Menter, F. 1993. Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows. AIAA. 93-2906.

[10] ANSYS 2017. ANSYS Fluent Theory Guide. Release 17.0. ANSYS, Inc.

[11] Adam J. “Proceedings of the 4th International Conference on Fluid Flow, Heat and Mass Transfer” (FFHMT’17), Toronto, Canada - August 21 - 23, 2017, Paper No. 170.

[12] Hariharan, P., Azar, J. 1996. PDC bit hydraulics design, profile are key to reducing balling. Oil and Gas Journal, 94(50), 58-63.