CSCanada
Advances in Petroleum Exploration and Development
Vol. 10, No. 2, 2015, pp. 103-107
DOI:10.3968/7925

ISSN 1925-542X [Print]
ISSN 1925-5438 [Online]
www.cscanada.net
www.cscanada.org

Research on the Bottom Hole Flow Field of PDC Bit

ZHANG Hui[a],*

[a] Drilling Technology Research Institute, Shengli Petroleum Engineering Co., Ltd, Sinopec, Dongying, China.
*Corresponding author.

Received 16 November 2015; accepted 21 December 2015
Published online 31 December 2015

Abstract
The three-dimension model of PDC bit was constructed in ANSYS software in order to obtain the physical model of the PDC bit's flow field on bottom hole. 3D geometric model of PDC bit was created by ANSYS and meshed by GAMBIT, analyzed by FLUENT by setting boundary condition of velocity inlet and outflow. Finally, we get the flow field figure while the velocity is 232 m/s. The district where fluid speed is slow and there is reverse flow is small between borehole wall and the nozzle which is far from the center of bit. The district where fluid speed is slow and there is reverse flow is small between axes of bit and the nozzle which is close to the center of bit. If the nozzle velocity is different, the effect to clear and carry over cuttings will be more obvious. The speed of the nozzle which is close to the center of bit should be highest. The two nozzles adjacent to this nozzle should have higher speed.

Key words: PDC bit; Jets; Flow field; FLUENT; ANSYS

Zhang, H. (2015). Research on the bottom hole flow field of PDC bit. *Advances in Petroleum Exploration and Development*, *10*(2), 103-107. Available from: URL: http://www.cscanada.net/index.php/aped/article/view/7925
DOI: http://dx.doi.org/10.3968/7925

INTRODUCTION

Bottom hole flow field is the space to discovery hydraulic problem of bits directly[1], we can find the hydraulic problem of bits, improve nozzle combination and design junk slot by means of analyzing flowing phenomenon of flow field. The rock breaking mechanism and cuttings removal mechanism of PDC bit is different from roller bit, number of nozzles and layout schemes are also different, so we can't use roller bit's bottom hole flow field characteristics instead of PDC bit, only study on PDC bit's flow field characteristics on bottom hole to make sense of PDC bit with nozzle structure, nozzle testing, bottom hole over characteristic and bottom hole fluidic rule. Therefore, it is very necessary and urgent to analyze PDC bit's flow field on bottom hole. The model which study on PDC bit's flow field on bottom hole is quite simple at present, in this paper full size model of PDC bit was created by ANSYS, analyzed flow field on bottom hole by FLUENT, obtained a certain research results which have great engineering signification in guidance of configuration optimization of PDC bit.

1. PDC BIT'S HYDRAULIC CHARACTERISTICS

The PDC bit hydraulics can be divided into two aspects[2]: One is the bit hydraulic parameters, the other is the bit flow field on bottom hole and hydraulic structure. The PDC bit's hydraulic parameters mainly include hydraulic control parameters and measures which should be taken to improve the drilling rate and prolong the service life of bit, such as displacement, pump pressure, nozzle diameter, hydraulic horsepower and drilling impact force. The bit's flow field on bottom hole and the hydraulic structure is primarily a bit designer should adopt the structure form when design bit on the basis of studying the flow law of fluid flow on bottom hole and cleaning and cooling mechanism of fluid flow, in order to make bottom hole fluid and hydraulic energy distribution achieve optimal cleaning and cooling effect, such as the nozzle size, shape, location, spray angle and the shape of the flow channel and location, and so forth.

The PDC bit hydraulic structure study the flow law of liquid flow on bottom hole and the bit's surface, hydraulic energy distribution and liquid flow's cleaning and cooling mechanism, namely according to different mechanical structure of the bit (the structure form of mechanical cutting), equipped with bottom hole flow field and pressure field which produced during the research of different hydraulic structure form; then a good flow field on bottom hole can be found, which can match with some kind of mechanical structure, migrate the cuttings on bottom hole as soon as possible, arrive at borehole wall in the shortest path and lift into the annular space, according to the hydraulic action on cleaning and cooling effect of bit.

Because under the condition of bottom hole, the bit's surface structure is complex, there are a number of nozzles. In terms of PDC bit, there are cutting teeth, steel place, channel and the rotation of the bit[3]. Therefore, according to the present basic theory of fluid mechanics, using analytical methods can't describe flow problems under the condition of such complex structure. For the research of bottom hole flow field, the methods which usually used are using flow visualization experiment to describe and evaluate, according to a particular structure of bit. But this method is restricted by experimental conditions and instruments, generally the complete flow description cannot be given and the bit of different structure must make different bit model experiment, as the experimental conditions can't be changed flexible, leading the cost is expensive. The assumption of using the method of numerical calculation to solve the basic equations of fluid mechanics can be true, with the advent of high speed computer, at present the computational fluid dynamics has been widely used in various hydraulic calculation. Currently, the study of drilling flow field on bottom hole and description method is divided into flow visualization experiment method and numerical simulation method. The former is also divided into the dyeing liquid[4], pasting[5], painting[6] and tracer particle track method[4-6].

In recent years, with the development of computational fluid dynamics, there is a numerical simulation method to study flow field of the bit[7]. The main content is reproducing the physical picture of PDC bit bottom hole flow field on computer by solving 3D incompressible Navier-Stokes equation under the condition of PDC bit. Through this method, we can describe the three-dimensional flow field more comprehensive and change bit hydraulic structure conditions more flexible, provide convenient research means for optimal design of bit hydraulic structure. However, using this method, the computer should be required to have large storage speed. With the large computer appear and continuous improvement of the numerical method constantly, this method will be more broad prospects for development.

2. SETTING UP THE PHYSICAL MODEL OF THE PDC BIT'S FLOW FIELD ON BOTTOM HOLE AND THE CALCULATION MODEL

2.1 Setting up the Physical Model

This paper established the physical model and coordinate system of the PDC bit which have four blades and five nozzles, they are shown in Figure 1, 174 mm bit diameter,100 mm inner diameter, 16 mm nozzle diameter. The five nozzles position is shown in Figure 2: The X axis in the graph is to the left, Y is to upward, Z is perpendicular to the paper face. Among them, the angle between the first nozzle center's radius and the –X axis was 27°, meanwhile the first nozzle center's radius and the third nozzle about bit center symmetry. The angle between the second nozzle center's radius and the +Y axis was 27°, meanwhile the second nozzle center's radius and the fourth nozzle about bit center symmetry. The angle between the fifth nozzle center's radius and the –X axis was 15°. The distance between the center of nozzle exit 1-4 and the bit center is 55 mm, the angle between the center line of nozzle 1-4 and Z axis was 10°. The distance between the center of nozzle exit 5 and the bit center is 36 mm, the angle between the center line of nozzle 5 and Z axis was 7°.

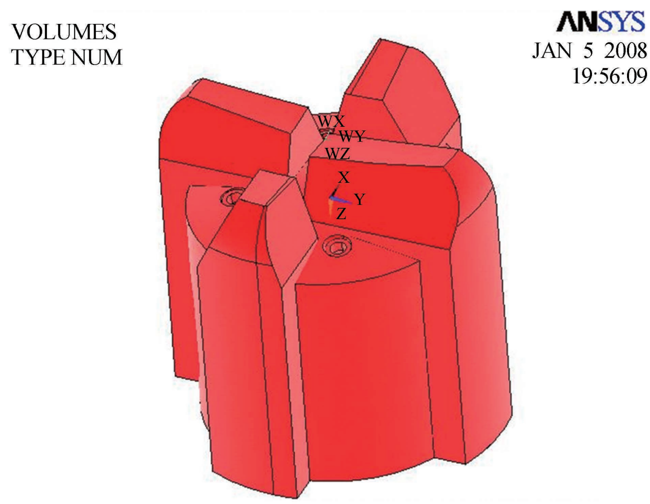

Figure 1
Block Map of Bit

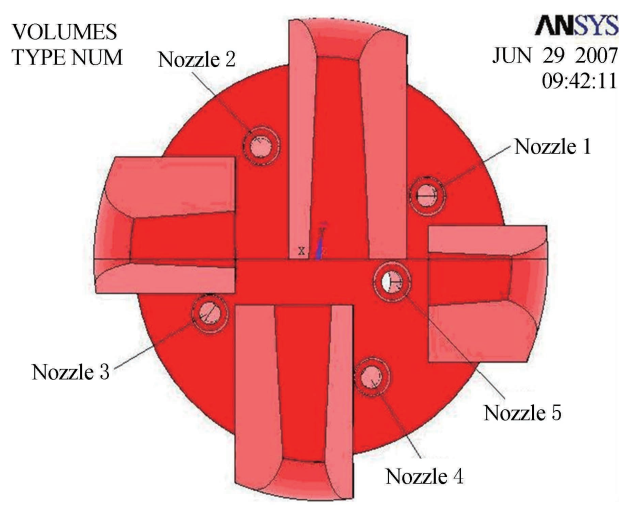

Figure 2
Position Diagram of the Five Nozzles

Although the bit model has been established, this model can't be used in FLUENT software analysis directly, because the route which fluid flow is entity in FLUNT software, so we need to make the model in turn, it means changing the present part of entity into space and changing the space part into entity. Figure 3 is the picture after the bit model is changed and divided grid in this way.

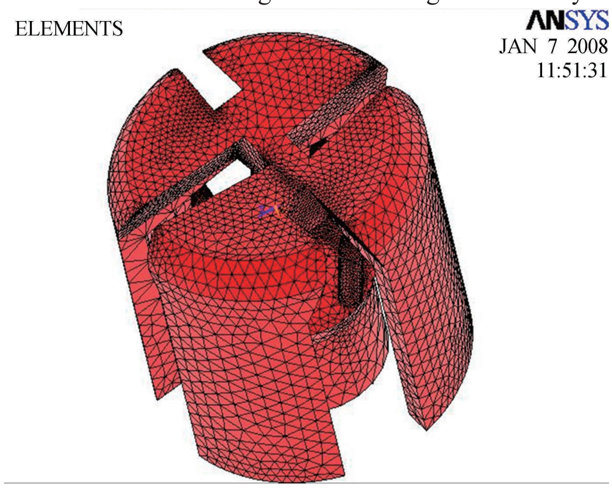

Figure 3
Grid Map of Physical Model

2.2 Establishing Calculation Model

The calculation model of the PDC bit bottom hole flow field is established by two steps. Firstly, the physical model which was created in ANSYS software should be meshed by GAMBIT, in order to obtain initial model. Secondly, establishing new model on the basis of the initial model.

2.2.1 Solvers and Setting up Boundary Conditions

We choose some kind of drilling fluid which the density is 1.3 g/ml and viscosity is 0.01 kg/m·s, set 3D unsteady flow, turbulence model is K-epsilon. The first round where the fluid enter bit is set up the velocity inlet, the velocity is 10 m/s; the four sides where the fluid return through annular space are set up outflow. The Turbulence Specification Method chooses Intensity and Hydraulic Diameter, Turbulence Intensity is 5%, Hydraulic Diameter is 100 mm.

2.2.2 Setting up Solutions

Firstly, initializing flow field, given the initialization is just a guess value on internal flow of, its data can be changed, and the result affects the convergence speed of iterative calculation. The convergence precision is set to 0.01, and then save the file.

3. CALCULATION RESULTS AND ANALYSIS OF PDC BIT FLOW FIELD ON BOTTOM HOLE

3.1 The Nozzle Flow Field Analysis

The relationship of five nozzles location has been given. Figures 4 and 5 are respectively the first nozzle's velocity vector and the velocity equivalent value line map.

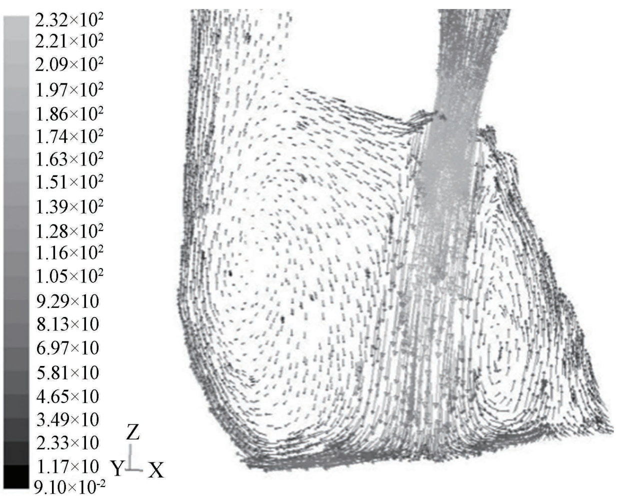

Figure 4
Velocity Vectors Picture of Nozzle 1

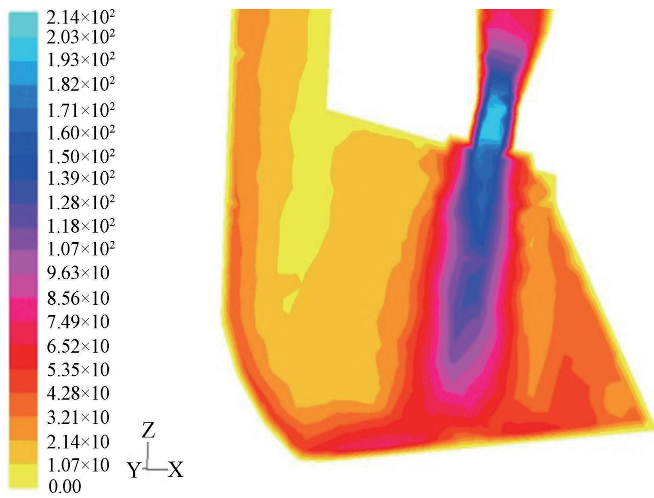

Figure 5
Velocity Contours Diagram of Nozzle 1

It can be seen from the figure that there has a vortex between the nozzle and borehole wall, the vortex is partial to one side borehole wall. The bottom of vortex is bottom hole, fluid flow in the direction of borehole wall by the nozzle center, the maximum velocity is 92.9 m/s, the minimum velocity is 34.9 m/s. The upper of vortex is the bit's cone, fluid flow in the direction of nozzle center by the borehole wall, the maximum velocity is 23.3 m/s, the minimum velocity is 8.5 m/s. Fluids spraying by nozzle shoot the bottom hole directly and then flow all around along the bottom hole, the maximum velocity is 209 m/s, the minimum velocity is 92.9 m/s. There also has a vortex between the nozzle and bit center. The bottom of vortex is bottom hole, fluid flow in the direction of bit center by the nozzle center, the maximum velocity is 81.3 m/s, the minimum velocity is 34.9 m/s. The upper of vortex is the bit's cone, fluid flow in the direction of nozzle center by the bit center, the maximum velocity is 34.9 m/s, the minimum velocity is 23.3 m/s.

3.1.1 Structure of Bottom Hole Flow Field

When the jet exit velocity is 232 m/s, the nozzle's velocity vector and the velocity equivalent value line map gives velocity vector and the velocity equivalent value line map through the nozzle jet axis of the cross-section. It can be seen from the figure that the structure of PDC bit's flow field on bottom hole can be divided into five regions: free jetting region, impinging region, cross-flowing region, radial flowing region and vortex region. The free jetting region is far away from the wall, the influence is very small, the flow shows the characteristics of free jet; the impinging region is near the wall, streamline is affected by bottom hole wall, curvature is bigger, it turned from the direction of the wall at an angle to the direction of the parallel to the wall, the flow velocity reduces quickly, pressure increases, forming a certain pressure gradient; the cross-flowing region is located between the bottom hole and the bottom of bit body, it is a flow region after where the jet leave the impinging region, almost parallel to the flow and bottom hole wall; the radial flowing region is the region that parts of liquids flow upwards along the wall after the bottom-hole cross flow arrived at the wall, it has an effect on carrying the cuttings from the bottom of the well; the vortex section is formed between the free jetting region and borehole wall as well as the wellbore axis and the free jetting section, due to the effect of bottom hole and borehole wall restriction and the influence of jet entrainment.

3.1.2 Velocity Field Analysis

It can be seen from the nozzle's velocity vector and the velocity equivalent value line map that the jet occurred violent momentum exchange and turbulent diffusion with water medium after it sprayed from nozzle, but there are still a part of the jets near the export keeping the nozzle exit velocity, the isokinetic nodal region is formed by this part of the jets. With the continuous advance of the jet, the isokinetic nodal region is end, leading to the radius of circular area and impact area increase, the velocity decay gradually along the axial and radial direction, but also the radial attenuation is faster than the axial attenuation. Five nozzle jets are within attenuation area before touching the bottom hole, this is beneficial to cleaning the bottom hole and carrying cuttings.

3.1.3 Cross Flowing Velocity Analysis

After the jet hit bottom hole, forming a thin layer with high lateral velocity in the space where is close to the bottom hole, and this is the cross flow layer, it is because of the cross flow, the cuttings of bottom hole can be cleared away in time, cleaning and cooling the bit cutting teeth.

3.1.4 Radial Flowing Velocity Analysis

The radial flowing region is located between the nozzle jetting region and the borehole wall, most of the fluid flow upward until the annulus exit. The general trend of velocity on annular radial flowing section is upward, but some velocity direction is oblique upward, leading to the general trend of radial flowing upward, at the same time forming the circular flowing. Few water flow down into the vortex section, which affected by the jet entrainment effect, this influence will reduce with the distance from radial flowing section to bottom hole, because of the bit flow model exit section is far enough from the bottom hole, the phenomenon of fluid reverse flow can't be seen in the annulus exit section.

3.1.5 Vortex Velocity Analysis

Within the scope of the borehole wall limited space, the high velocity of nozzle injection make the momentum exchange with the existing flooding fluid, its boundary layer increases continuously, forming the cross flow, which flow along the radial flow restricted by borehole wall and forced to turn to export, it drives the flooding fluid movement in this process, and the vortex region is formed.

3.2 Bottom Hole Flow Field Analysis

The flow field of bottom hole space and the nozzle flow field is also important, they all have an impact on drilling process directly, the velocity vectors picture of flow field is shown in Figure 6.

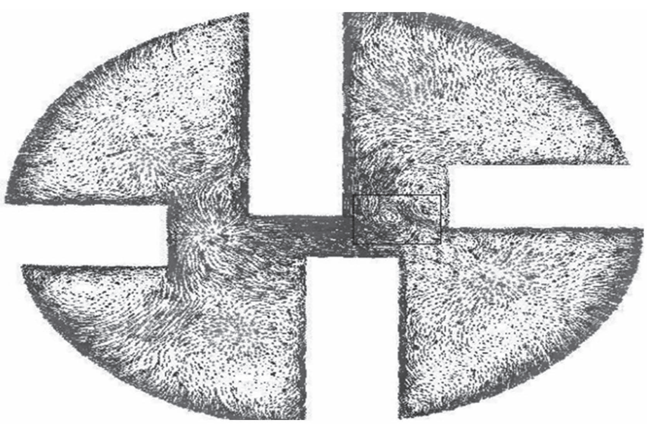

Figure 6
Velocity Vectors Picture of Flow Field

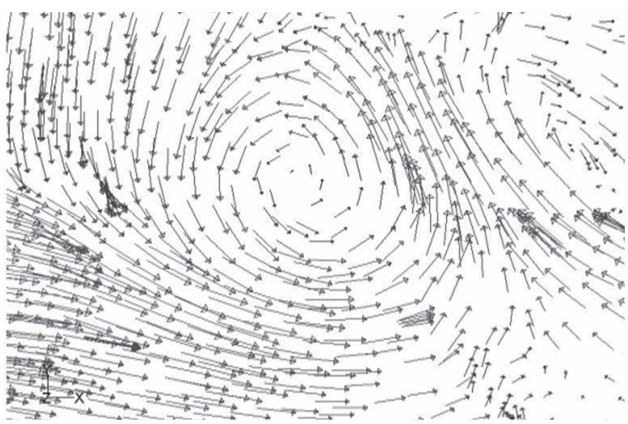

Figure 7
Expanded View of Swirl

It can be seen from the Figure 6 that the velocity in the center of the bottom hole is relatively fast, as well as the lifting velocity along the borehole wall around, it is beneficial to removing and carrying the cuttings in the center of bottom hole. But on the right side of the figure forming a vortex near the center of the bottom hole, the maximum velocity is 46.5 m/s. There is an obvious vortex in the lower left half part of the figure, the maximum velocity is 23.3 m/s, but the vortex have no obvious impacts on flow field and carrying cuttings.

As is showing in Figure 7, the maximum velocity is 46.5 m/s in the vortex, it is formed by the fluid of the center of bottom hole through nozzle 5 meet the jet of nozzle 2 and nozzle 1, which has a bad effect on carrying and removing cuttings. So one can see that the jet of nozzle 5 which is the latest from the center of bit has an effect on cross flowing field, which is beneficial to removing cuttings, the nozzle 1 and 4 is located at two sides of nozzle 5, its jet can be combined to one place with the jet of nozzle 5, it will have faster jet velocity, thus it can be more effective to remove and carry cuttings. When the jet of nozzle 2 and nozzle 3 collides with the jet of nozzle 5, it is easy to form reflux region, so the nozzle 2 and nozzle 3 can has different velocity or different distances from bottom hole, reducing the possibility of forming reflux region, more beneficial to remove and carry bottom hole cuttings.

CONCLUSION

In this paper, a three-dimension model of PDC bit was created by ANSYS software, analyzed the flow field by FLUENT under the condition of the bit is stationary, and conclusions are drawn as follows:

(a) The structure of PDC bit's flow field on bottom hole can be divided into five regions: free jetting region, impinging region, cross-flowing region, radial flowing region and vortex region.

(b) The vortex region between borehole wall and the nozzle jet which is far from the center of bit and jet angle is big is small, which is beneficial to carry and remove cuttings near borehole wall. The vortex region between drill axis and the nozzle jet which is close to the center of bit and jet angle is small is small, which is beneficial to remove cuttings of bottom hole.

(c) If the nozzle velocity is different, the effect to clear and carry over cuttings will be more obvious. The speed of the nozzle which is close to the center of bit should be highest. The two nozzles adjacent to this nozzle should have higher speed.

(d) It is more easy to form reverse flow when the fluid between the nozzle where close to the center of the bit and its interval nozzle met.

REFERENCES

[1] Sun, M. G. (2006). Design and field test of new PDC bit. *Oil Drilling Technology*, *28*(2), 21-24.

[2] Guan, Z. C., Chen, T. G., & Liu, X. S. (1996). Experimental study on the characteristics of fluid distribution at bottom hole of PDC bits. *Journal of the University of Petroleum (Edition of Natural Science)*, *20*(3), 24-27.

[3] Guan, Z. C., Chen, T. G., & Liu, X. S. (1994). Review of studies of PDC bits hydraulic configurations. *Journal of the University of Petroleum (Edition of Natural Science)*, *18*(6), 136-142.

[4] Glowka, D. (1983). Optimization of bit hydraulic configurations. Glowka, D. (1983). *Society of Petroleum Engineers Journal*, *23*(1), 21-32.

[5] Zhang, S. H. (1981). Study on bottom hole flow field of jet drilling. *Oil Drilling Technology*, (1), 25-28.

[6] Bission, P., Choux, J. C., & Provo, A. (1988, October). *Hydraulic optimization of PDC bits by visualization methods*. Paper presented at European Petroleum Conference, London, United Kingdom.

[7] Hu, J., & Yang, Z. F. (2011). Numerical simulation of the bottom hole flow field of PDC bit. *Journal of Yangtze University (Natual Science Edition)*, *11*(8), 41-43.

