# Flow Field Analysis of Turbine Blade Modeling Based on CFX-Blade Gen

Liguang Wang[1, 2, a, *], Zhijun Li[1, 2, b], Yu Wang[1, 2, c], Baolin Liu[1, 2], Qin Zhou[1, 2]

[1]School of Engineering and Technology, China University of Geosciences, BeiJing 100083, China

[2]Key laboratory on Deep Geo-Drilling Technology of the Ministry of Land and Resources, BeiJing 100083, China

[a]1054203098@ qq.com, [b]389345024@ qq.com, [c]wangyu203@cugb.edu.cn

**Abstract.** Turbodrill is an all-metal downhole drilling motor, whose turbine blades is the most important hydraulic components. As a result, it is of great importance to analyze impact of the drilling fluid on the blade flow field, so as to effectively guide the design of hydraulic components .Based on Bladegen software and ANSYS/CFX software to make simulation analysis on stator and rotor blades, the corresponding flow field of pressure and velocity are obtained. The relationship between the flow field and the geometrical parameters of blade is studied through flow field analysis, at the same time, it provides a technical means to predict the hydraulic performance of the whole turbine blades.

**Keywords:** Turbodrill, The stator and rotor, CFX-Bladegen, Flow field analysis.

## 1. Introduction

All-metal downhole motor turbodrill has a characteristic of high temperature resisting, and can be used for hard strata drilling together with impregnated diamond bit. It is an indispensable tool in ultra deep hole drilling with high temperature and pressure. Especially turbodrill of small diameter has very good prospects in exploration and development such as impregnated diamond bit drilling,coiled tubing drilling and directional drilling of unconventional energy sources etc[1].The most important hydraulic design components of the turbodrill are the stator and rotor blades , the hydraulic performance of which determine characteristics of the whole turbodrill. As a result, it is important to research value and significance to study the flow field of the blades. In this paper, we make full use of CFX-BladeGen software and ANSYS/CFD simulation software to simulate the flow field impact of drilling fluid on the stator and rotor blades of Ø89mm turbodril [2]. Through the analysis of the flow field of the blade, the relationship between the blade flow field and the geometric parameters of the blades is studied, so as to shorten research time and also reduce the times of bench experiment, providing a technical support for predicting the hydraulic performance of the whole turbine blades.

## 2. Φ89mm of turbodrill blade design theory

### 2.1 The working principle of turbodrill

Turbodrill transforms the drilling fluid pressure energy and kinetic energy through the turbine blades into the output shaft of the mechanical energy to drive the drill continuous rotation to break rock [3].

Table 1 The Φ89 mm turbodrill design parameters

| Name | Numerical and units | Name | Numerical and units |
|---|---|---|---|
| Outer diameter | 89mm | Power | 20~40kW |
| Flow rate | 6L/s | Pressure drop | 4~8MPa |
| Rotation | 1500~2000r/min | Rated torque | 800~2000N m |
| Fluid density | 1000~2000kg m-3 | Brake torque | 300 N.m |

According to field experience of the geological drilling, design parameters of theΦ89mm turbodrill are selected and shown in table 1.

### 2.2 Design methods of the blade

At present, there are a lot of blade modeling methods. The conventional curve modeling is basically no longer use combined with continuous three order derivatives, and does not meet the blade outlet angle and blade runner, continuous contraction [4]. The current modeling of turbine blade has two categories: modeling of geometric the parameters and modeling based on software platform.

Table 2 Classification of turbine blade modeling methods

| Shape classification | Modeling methods | Advantages and disadvantages |
|---|---|---|
| parameters of geometric modeling | Four order spline | Advantages: With a continuous three order derivatives, meet the requirements of the blade outlet angle, blade runner and continuous contraction. Disadvantages: Complex calculation process, requires the use of mathematical software. |
| | Three polynomial | |
| | Five polynomial | |
| | Bezier curve | |
| Based on software platform modeling | Bladegen and Matlab programming geometric modeling | Advantages: The blade line generates smooth and fast effectively. Disadvantages: Parameters are more and complex. |

## 3. The simulation process of ANSYS/CFX

At present, CFX has been widely used in the aerospace, rotating machinery, energy, petrochemical industry, machinery manufacturing, automotive, biotechnology, and water treatment [5]. CFX, including BladeGen, TurboGrid, CFX Pre, CFX Solver Manager and CFX Post, provides customers with a more specialized tool for design and analysis of rotating machines in the field of rotating machinery.

### 3.1 CFX-BladeGen

BladeGen requires users to provide several geometrical parameters: inlet Angle and outlet Angle, installation angle of blade, number of blade, blade wheel hub and outer data. We can observe the flow and the blade of the stator and rotor through the two-dimensional meridian plane window [6].

Input the geometric parameters which have been identified in the window of the meridian plane and the blade parameter separately. In order to better determine the blade upstream and downstream boundaries in the window of meridian plane flow channel, it needs to extend some distance to the upstream and downstream at the entrance of the stator blade and rotor exit respectively. The meridian plane and model of the stator and the rotor have been shown in Fig.1, 2.

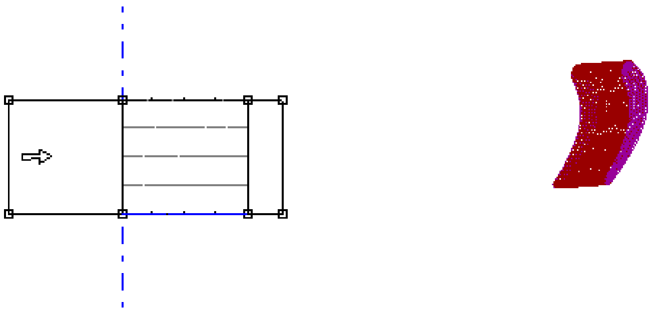

Fig.1 Two-dimensional meridian plane and model of the rotor

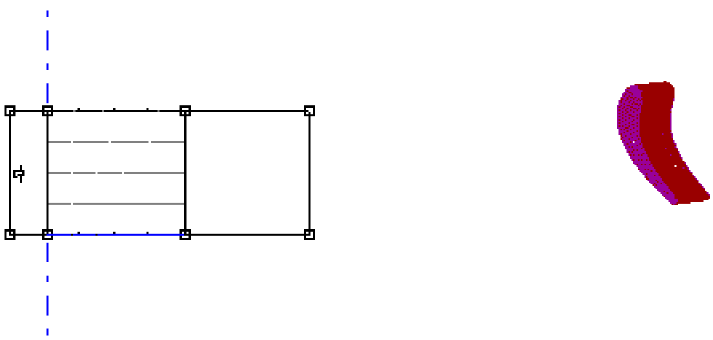

Fig.2 Two-dimensional meridian plane and model of the stator

### 3.2 CFX-Turbo Grid

By using the workbench design modeler meshing step, a mesh system is created over the geometry constructed in the initial step. ANSYS CFX lets the user control mesh density, cell size, cell distribution in local and global parts and cell shapes.

Turbo Grid generates tools for the grid cascade passage. It adopts innovative grid template technology, combined with parametric ability, can generate high quality grid cascade passage for the vast majority of blade type quickly and simply. Configuration data file only needs to provide the number of blades, blade and hub and the outer cover, can complete the stator and rotor flow grid. The mesh generation of the stator and rotor was shown in Fig.3.


Fig.3 Mesh generation of the stator and rotor

### 3.3 CFX-Pre

After importing port grid and rotor blade model respectively, CFX-Pre needs to select special rotary mechanical module, and physical definition. The blade passage domain, the physical model, the fluid type and boundary conditions should be defined. After defining the physical model, pretreatment model is shown in Fig.4:

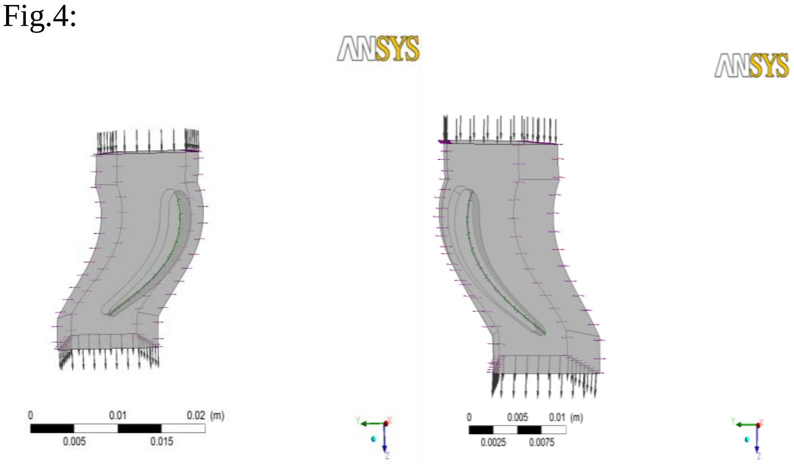


Fig.4 Pretreatment model of blade

### 3.4 CFX-Solver and CFX-Post

The use of CFX Solver Manager sets the solver options, through the batch mode iterative method for solving the control equation and the convergence is reached. Choose the most robust convergence boundary definition: inlet is set the flow rate, outlet is set the pressure value. TurboPost is included in CFX Post and can be output in the form of the flow field contours as shown in Fig.5, Fig.6 and Fig.7.

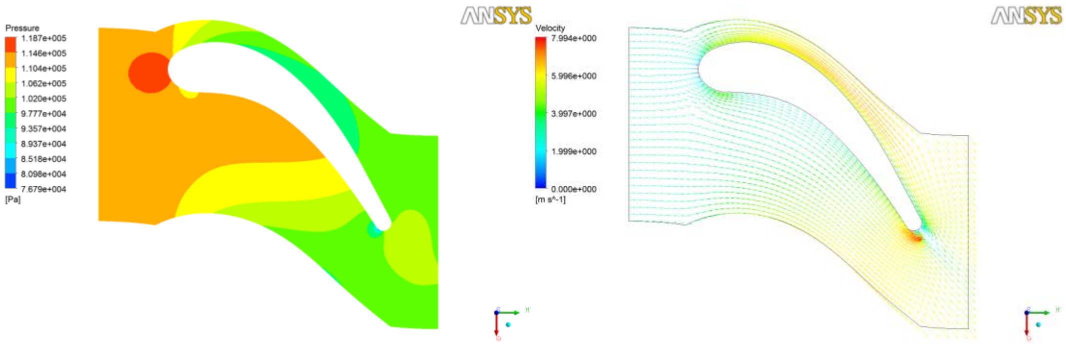


Fig.5 Pressure and velocity flow field of the stator

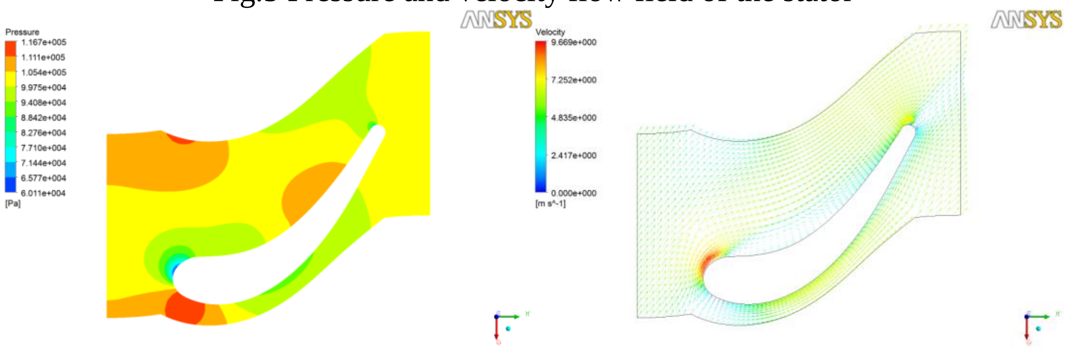


Fig.6 Pressure and velocity flow field of the rotor

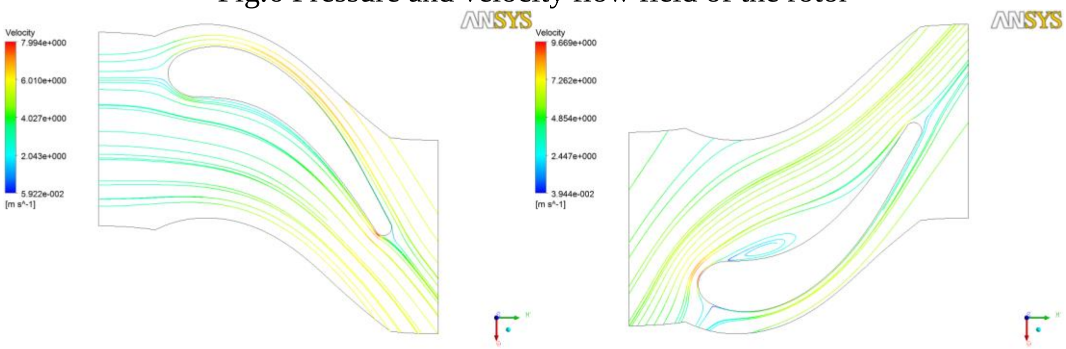


Fig.7 Streamline of the stator and rotor

The results of flow field analysis:

1. From the stator, the rotor blade surface to the outlet end pressure field is a gradient array. The suction surface of blade leading edge turning is greater than leaves. The gradient variation of the pressure can explain the size of the output torque.

2. With drilling fluid impacting on the turbine stator blade pressure and suction surface, the flow along the blade passage can accelerate and reaches its maximum of 7.99 m/s when through the stator blade trailing edge. Stator leading edge rate is almost zero, meanwhile, no vortex flow is found in the streamline distribution, so the cotyledon mold design is reasonable.

3. With drilling fluid flowing into the turbine rotor blades, it can impact the pressure surface and the suction surface, and can accelerate along the blade passage .The speed of rotor leading edge can reach its maximum of 9.67 m/s . Whirlpool is found in the streamline distribution shown in Fig.7, so the rotor blade design is not reasonable.

## 4. Conclusion

Based on Blade-Gen software and through use of ANSYS/CFX software to make simulation analysis on stator and rotor blades, the corresponding pressure and velocity vector of flow field are obtained. The study shows that velocity of drilling fluid changes a little after it enters the suction surface of the turbine stator blade, but its trailing-edge velocity has a greater increase. Therefore, the stator blade design is reasonable; the leading-edge velocity of the flow fluid in the rotor passage is so large that it is easy to create whirlpools on suction surface, resulting in greater hydraulics lose.

Therefore, the rotor blade curve designing is unreasonable. So the geometrical parameters need to be revised. The relationship between the flow field and the geometrical parameters of blade is studied through flow field analysis, meanwhile, it provides technical method to predict the hydraulic performance of the whole turbine blades.

## Acknowledgments

This work is supported by the Fundamental Research Funds for the Central Universities (No. 2652015061) and Beijing Organization Department Outstanding Talented Person Project (No.2013D009015000002), the Beijing Higher Education Young Elite Teacher Project (Grant No. YETP0645). Meanwhile, great thanks also go to former researchers for their excellent works, which give great help for our academic study.

## References

[1] Lin Yuanhua et al: New turbodrill blade type line and computer aided design,Journal of Chongqing University, Vol.27(2004)No.2,p.24-39.
[2] Ding Lingyun, Feng Jin: CFX-BladeGen in the turbine blade modeling application,Journal of Engineering Design, Vol.12(2005) No.2,p.36-59.
[3] Wang Yu, Yao Jianyi, and Li Zhijun: Design and Development of Turbodrill Blade Used in Crystallized Section,The Scientific World Journal, Vol.2014,Article ID 682963, p.1–12.
[4] A. Mokaramian, V. Rasouli, and G. Cavanough: Fluid flow investigation through small turbodrill for optimal performance,Mechanical Engineering Research, Vol. 3(2013)No.1,p.1–24.
[5] S.N.Singh,V.Seshadri,R.K.Singh,andT.Mishra: Flow characteristics of an annular gas turbine combustor model for reacting flows using CFD, Journal of Scientific and Industrial Research, Vol. 65(2006) No.11, p.921–934.
[6] Q. Zhang, M. Li, Z. Cheng et al: Development and application of test bed for large torque screw drill,China Petroleum Machinery, Vol.35(2007) No.7, p.31–34.