Flow Field Analysis and Performance Prediction of the Φ127 Turbodrill

Liguang Wang [1, 2, a,*], Zhijun Li[1, 2, b], Yu Wang [1, 2], 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

Keywords: Turbodrill, Blade, Flow field, Performance prediction.

Abstract. The most important part of the hydraulic design part in turbodrill is stator and rotor blades. The hydraulic performance determines the output characteristics of the turbodrill. SolidWorks three-dimensional (3D) modeling is applied to Φ127 turbodrill blades. Simulation analysis is using the Software ANSYS/CFD to make, the corresponding pressure and velocity flow field are obtained. Finally, the output performance of the turbine section is predicted through repeated simulation of different speed flow field. The predictive results show that with clean water as the fluid medium and the flow rate of 15 L/s of the turbine section, its maximum power point and the braking torque can reach10 kW and 325N•m respectively with optimal working speed of 275 r/ min. Therefore, it is a reasonable design that meets all requirements.

Introduction

As an all-metal downhole motor turbodrill, it has a high temperature resisting characteristic, and can be used for hard strata drilling together with impregnated diamond bit. It is an indispensable tool in ultra deephole drilling with high temperature and pressure.Meantime,turbodrill has very good prospects in exploration and development such as high temperature hot dry rock drilling, coiled tubing drilling and directional drilling of unconventional energy sources etc [1]. The most important hydraulic design parts of the turbodrill are the stator blades and the rotor blades, the hydraulic performance of which determine output characteristics of the drilling. As a result, it is of important study value and research significance to predict hydraulic performance of the rotor blades [2].Based on Φ127 turbodrill blades, the paper makes full use of 3D modeling software and ANSYS /CFD simulation software to simulate the pressure and flow field speed of the blade, and predicts the output characteristics of the turbine section through flow field analysis at different rotating speed.

Φ127 Turbodrill Blade Design

Working mechanism and design goals of Φ127 mm turbodrill.

Table 1 The Φ127 mm turbodrill design parameters

Name	Numerical and units	Name	Numerical and units
Outer diameter	127mm	Power	10~20kW
Flow rate	15L/s	Pressure drop	4MPa
Rotation	200~500r/min	Rated torque	1000~2000N·m
Fluid density	1000~2000kg/m-3	Brake torque	400N·m

Turbodrill can turn pressure energy and kinetic energy of the drilling fluid into mechanical energy of rotation of the output shaft through hundreds of rotor blades, so as to drive a continuous rotation of the drill bit, achieving the purpose of crushing rock. In oil drilling field, the stator and rotor of turbodrill are made to be axial- flow, so that the drilling fluid can flow along the guide vane [3]. According to the Chinese continental scientific drilling data, the rock is mainly composed of eclogite when the drilling depth is more than 5000m [4]. Combined with the design requirements of ten thousand meters deep, the turbodrill design goals of Φ127 mm is shown in table 1:

Φ127 turbodrill blade modeling.

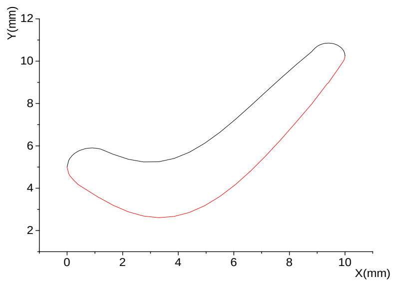


Fig. 1 Curve of the blade

The turbine has 24 blades, with a Turbine rotor blade height of11mm, a blade pitch of 10.99, and an installation angle of 68 degrees. Because the stator and rotor are mirror symmetry, the export structure of rotor blades is equal to the import structure angle of the stator blades [5]. The shaping of blade uses five order polynomial curve modeling. The blade curve is shown in Fig 1.

The ANSYS CFD simulation

CFD Model Design and Meshing generation.

After the model is drawn through the SolidWorks software and imported into ANSYS, a single stage turbine rotor flow calculation model is created, as is shown in Fig 2. In order to improve the precision of computer and get the stable convergence of the flow field, it needs to extend a certain distance in the entrance of stator and the outlet of rotor separately.

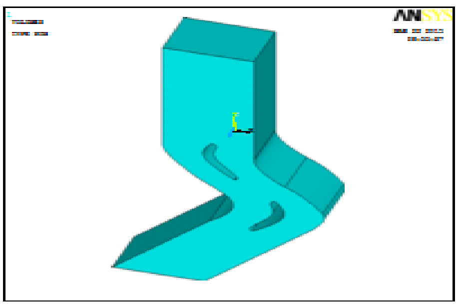


Fig. 2 Single-period model

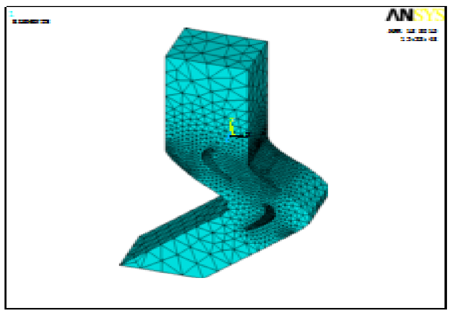


Fig.3 Meshing generation

Flotran CFD module in ANSYS is chosen after the model body was formed and three dimension Flotran142 units are added at the same time. We applied meshing free method in order to get more accurate results of distribution of flow field in the simulation [6]. As is shown in Fig 3, the meshing method offered the partial encryption processing in the junction of stator and rotor.

Boundary conditions and CFD solver.

Boundary conditions are imposed in the process of ANSYS simulation. The simulation process of the blade is defined as follows:

(1) The positive Y-axis represents the direction of rotation; while the negative represents the flow direction of the drilling fluid.

(2) The cross section(a gap in fact) surface of the stator and rotor is defined as frozen rotor; the boundary of the stator and rotor is defined as a periodic cycle.

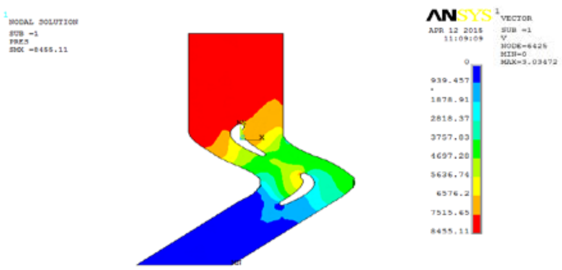
(3) The blade surface, the leading edge, trailing edge of the stator and rotor rotates with the rotor rotation are defined as no slip wall boundary.

(4) The model of drilling fluid in terms of temperature is set to be adiabatic; the Turbulence model adopts the standard $k-\varepsilon$ model which is widely used in engineering.

(5) Turbulence equation uses the wind convection item format, the others set to the default values.

Inlet sets fluid flow, outlet sets pressure, and initial speed value is input. In a broad view of a CFD project, convergence, convergence criteria or tolerance values attained for diverse equations, residuals, stability, under-relaxation factors. The computer starts running after performing Solving operation.

The simulation analysis of flow field.



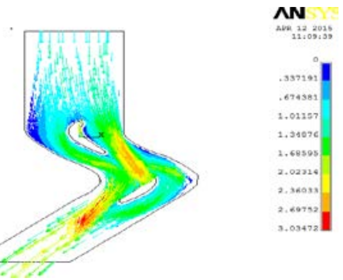
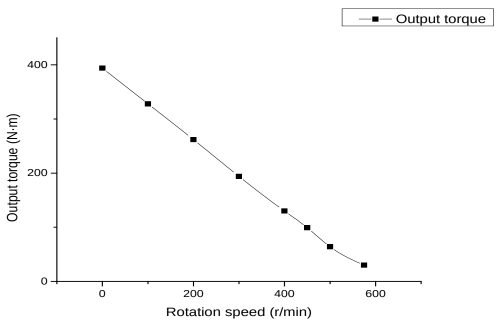


Fig. 4 Flow field of velocity Fig. 5 Flow field of pressure

In the Fig 4 above, the variation of fluid pressure distribution is clearly presented.The results show that the pressure was significantly greater in the pressure surface than that in the suction surface, so it guides the rotor rotating, and produces the torque.In the Fig 5 below, the variation of fluid velocity distribution is clearly presented. When the drilling fluid flows into the stator,one part impacts the suction side of the stator,another part impacts the pressure surface. After the fluid enters the stator blade, it is found that the change of fluid velocity is not obvious. The fluid velocity which is on the both side of the stator trailing accelerates gradually. The fluid velocity approximates to zero at the end of trailing .When the drilling fluid flows into the rotor, the fluid velocity of suction surface is lower than the pressure surface, meanwhile, the fluid velocity of pressure surface increases gradually, and reached the maximum speed in the trailing edge.

Performance prediction

Combined with CFD analysis, according to the above simulation can simulate the torque and power of different speed under the condition of drilling fluid density is $1\,g/cm^3$ (the fluid medium is water) and the flow of 15 L/s. Then the single-stage torque and power of turbine blade translates into the class of 100 set the torque and power of the turbine section of turbine blades. Finally, the output of the turbine section characteristics is obtained. Assume that all levels of the turbine under the working condition of boundary conditions and so on all is consistent, its output torque and power variation with the rotational speed as shown in Fig 7 and Fig 8.



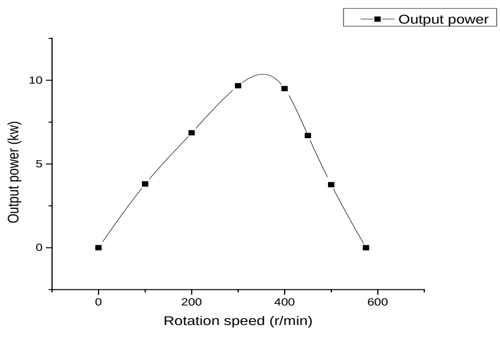


Fig. 6 The output power curve of simulation Fig. 7 The output torque curve of simulation

Fig 6 represents the output torque trends with the increasing rotation speed of the rotor. The curves shows that the output torque is a linear decreasing trend with the speed increasing of the rotor. Fig 7 represents the output power trends with the increasing rotation speed of the rotor. The curves shows that the output power is a parabolic trend with the speed increasing of the rotor.By using the water as the drilling fluid, the simulation results show that the output torque has obvious linear relationship with the rotational speed under the condition of the flow of 15 L/s. With the increase of the rotational speed, the torque gradually becomes smaller. As the rotational speed increases, power decreases after increasing first, consistent with the theoretical characteristic curve of turbodrill. The maximal output power and the optimal rotational speed of the turbine section is 10KW and 275rpm, respectively. The turbine section of the braking torque is up to 394 N· m, at the same time, the working torque can reach 325N·m.The simulation results are equal to the design goals.

Conclusions

Through the analysis of the blade pressure and velocity of flow field, it predicts the output performance of the turbine section. Conclusions are as follows:

1) The flow field of stator and rotor suction surface have high velocity and small pressure, but in the pressure surface have low velocity and big pressure, so produce a pressure difference between suction surface and pressure surface which makes the rotor rotation.

2) Under the condition of a certain flow, the output torque increases with the speed increasing and the turbine power variation shows a parabolic trend.

3) According to the simulation results, Φ127 turbodrill blades satisfy the requirements of drilling tools of torque and rotational speed.

Acknowledgments

The authors gratefully acknowledge the support by the International Scientific and Technological Cooperation Projects (Grants nos.2010DFR70920 and 2011DFR71170), the Fundamental Research Funds for the Central Universities (No. 2652015061, 2652015059) and the Beijing Higher Education Young Elite Teacher Project (Grant no.YETP0645).

References

[1] Zhao Hongbo,Liu Baolin,Wang Jianqiang.Turbodrill blade type line design and the flow field simulation analysis research.J-rock and Soil Drilling and Digging Engineering Prospecting Engineering.Vol.39(2012)No.11, p.29-32.

[2] Feng Jin,Fu DaLiang.Turbodrill turbine blade design new method.Journal of Petroleum Machinery. Vol.28(2000)No.11, p. 9-12.

[3] 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.

[4] A.Mokaramian,V. Rasouli,G.Cavanough.Adapting oil and gas downhole motors for deep mineral exploration drilling.The 6th International Seminar on Deep and High Stress Mining. Perth,Australia,2012.

[5] S.N.Singh,V.Seshadri,R.K.Singh, et al. 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] Ding Lingyun,Feng Jin.CFX-BladeGen in the turbine blade modeling application,Journal of Engineering Design.Vol.12(2005) No.2,p.36-59.