Advanced Science and Technology Letters
Vol.63 (NGCIT 2014), pp.48-52
http://dx.doi.org/10.14257/astl.2014.63.10

Numerical Simulation of Downhole Completion Equipment via Computational Fluid Dynamics

Chao Zheng, Yonghong Liu*, Hanxiang Wang, Renjie Ji, Zengkai Liu

(College of Mechanical and Electronic Engineering, China University of Petroleum, Qingdao, Shandong, 266580, China)
{zhengchaoupc, liuyhupc, wanghx1899, jirenjie, liuzengk}@163.com

Abstract. In the past decade, hydraulic fracturing technology has been widely used to improve the permeability and production of reservoirs. During the fracturing process, fracturing sleeves were installed and run underground with the work condition is harsh and unpredictable. In this paper, the internal flow status of fracturing sleeve is investigated using a coupled approach of Computational Fluid Dynamics and Discrete Element Method. Based on this study, the internal flow characteristics of fracturing fluid and sand particles are obtained. The simulation results show that the vortex exists at the port area. Velocity and pressure gradients change dramatically at the port due to flow filed becoming narrower. The results also indicate that the Computational Fluid Dynamics is an effective tool to study the dynamics and internal flow of the fracturing tool.

Keywords: Internal flow; Hydraulic fracturing; Computational Fluid Dynamics

1 Introduction

Hydraulic fracturing is a method which injects water, sand and chemicals deeply into the ground to break up the formation and release oil & gas [1]. During the fracturing process, the fracturing fluid containing sand particles as proppant is pumped underground [2]. As it is known, fracturing process has a range of features such as high pressure, high sand rate and high flow rate. The fracturing sleeve which acts as key component is mounted in the well. The structure and parameter of fracturing sleeve has a significant influence on the internal flow status. The inappropriate design of fracturing tool may lead to the serious vortex phenomenon which is harmful to the usage.

To get a better understand of fracturing process, Computational Fluid Dynamics is used to calculate the motion of fracturing fluid and sand particles. According to previous references, a lot of researches have been conducted on the flow analysis in other industrials using the CFD approach [3-4]. The coupled approach of Computational Fluid Dynamics and Discrete Element Method is widely used to simulate the solid-liquid two phase flow [5].

This paper presents a CFD simulation of the fracturing sleeve. The motion of fracturing fluid and sand particles is obtained based on the CFD model. The remainder of the paper is presented as follows. Section 2 establishes the simulation

ISSN: 2287-1233 ASTL
Copyright © 2014 SERSC

model. Section 3 is the simulation results and discussion. Section 4 summarizes the paper.

2 Numerical Simulation

2.1 Model Establishment

According to the actual structure of fracturing sleeve, geometry is established and meshed in the Gambit. Fig. 1 presents the mesh result of fracturing sleeve geometry. In the fracturing process, the fracturing fluid flows into the sleeve from the inlet and flows out from the outlet. The wall surrounds and restraints flow of fracturing fluid. The ports are the channel connecting the inner area and annulus which forms between fracturing sleeve and casing. The flow flied narrows at the port area, which will lead to the change of flow status.

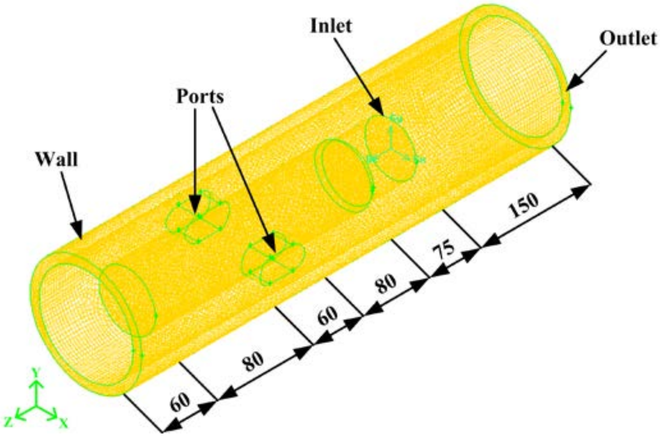

Fig.1. Mesh generation of the fracturing sleeve geometry.

2.2 Calculation Condition

To get accurate simulation result, calculation condition must be given correctly. According to actual fracturing process, the calculation condition is listed in Table 1. These parameters such as flow rate, density and viscosity have an important influence on the flow status of fracturing fluid. Parameters listed in the table provide the initial conditions for computational equations such as turbulence equation, force equation and energy equation. In the calculation, the equations solve the movement problem of fracturing tool and sand particles. When the calculation is finished, simulation results such as velocity, pressure and trajectory can be obtained.

Copyright © 2014 SERSC

Table 1. Calculation condition.

Parameter	Value	Unit
Flow rate	4.5	m^3/min
Density of liquid	1100	kg/m^3
Viscosity of liquid	100	mPa·s
Density of sand	2300	kg/m^3
Diameter of sand	380	μm
Sand flow rate	12	kg/s

3 Simulation Results and Discussion

Fig. 2 shows the velocity vector of plane Y=0. It is clearly to observe that velocity is small and uniform at the entrance, but large at the port. It indicates that vortex and backflow which are harmful will occur at the port. Moreover, the velocity of fracturing fluid at the entrance is nearly 20m/s, while the velocity can reach up to 63m/s at the port. The change of velocity attributes to the change of flow flied.

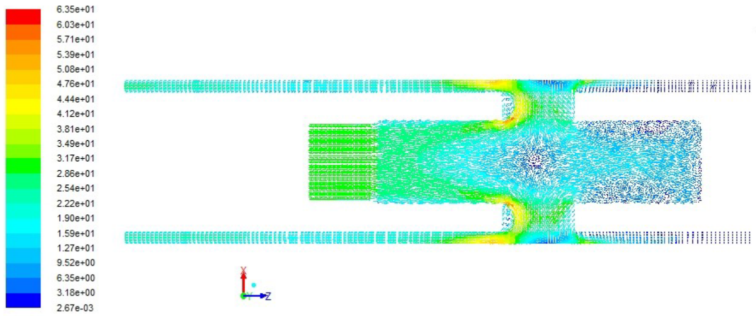

Fig. 2. Velocity vector of plane (Y=0).

Fig. 3 shows the pressure contour of plane Y=0. From the Figure, it can be seen that the change trend of pressure is similar to velocity. The pressure gradient changes dramatically at the port area. Pressure is found to reduce from the entrance to exit. The factors contributing to pressure loss include the friction between fracturing fluid and inner surface, the collisions between particles and wall surface and the inner friction of fracturing fluid. The interaction of these factors contributes to the reduction of pressure. The pressure loss may affect the fracturing result. In the design of fracturing sleeve, methods are applied to decrease the pressure loss.

 Copyright © 2014 SERSC

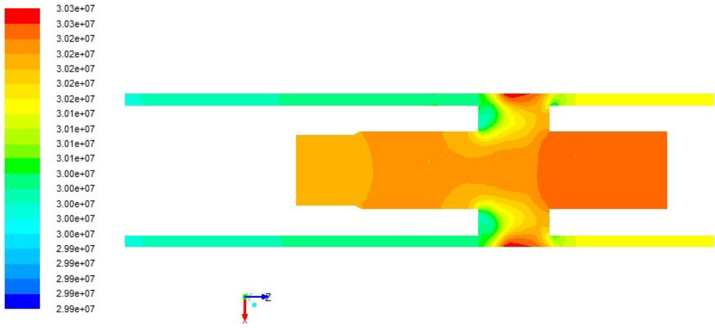

Fig. 3. Pressure contour of plane (Y=0).

Fig. 4 shows the trajectories of sand particles. In the fracturing process, sand particles are carried by the fracturing fluid. From the Figure, fluctuations and uncertainty of motion trajectories are obvious. This may be explained that sand particles are affected by the various forces such as Gravity force, Drag force, Pressure gradient force, Buoyancy force and Added mass force.

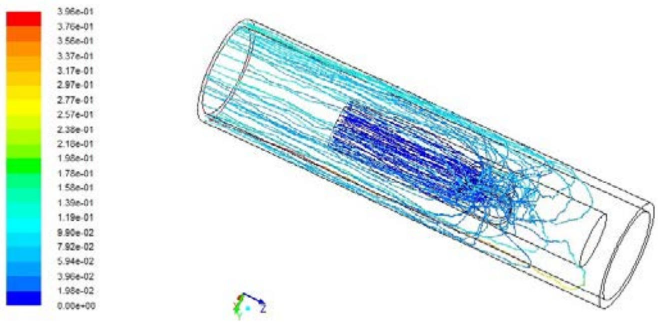

Fig. 4. Trajectories of sand particles.

4 Conclusions

(1) This paper presents a CFD study of fracturing tool including the internal flow characteristics of fracturing fluid and the motion of sand particles.

(2) Referring to the simulation results, large velocity and pressure gradient generates at the port area due to the narrowing of flow filed. The maximum velocity of fracturing fluid is located at the port.

(3) Based on the analysis of flow status, the high erosion rate area can be identified. It indicates that CFD approach is an effective tool to simulate the flow status of fracturing sleeve. The simulation results obtained can be instructive to the fracturing tool design.

Copyright © 2014 SERSC

Acknowledgements. The authors gratefully acknowledge the support provided by the National Natural Science Foundation of China (No. 51205411), the Shandong Provincial Natural Science Foundation of China (No. ZR2012EEL15), National High-Technology Research and Development Program of China (No.2013AA09A220), and Taishan Scholar project of Shandong Province (TS20110823).

References

1. Rutqvist, J., Rinaldi, A.P., Cappa, F.: Modeling of fault reactivation and induced seismicity during hydraulic fracturing of shale-gas reservoirs. J. Pet. Sci. Eng. 107, 31--44 (2013)
2. Mahrer, K.D.: A review and perspective on far-field hydraulic fracture geometry studies. J. Pet. Sci. Eng. 24, 13--28 (1999)
3. Atkinson, M., Stepanov, E.V.: High pressure testing sand erosion in 3D flow channels and correlation with CFD. Wear. 263, 270--277 (2007)
4. Maurer, C., Schulz, U.: Solid particle erosion of thick PVD coatings on CFRP. Wear. 317, 246--253 (2014)
5. Zhang, M.H., Chua, K.W., Wei, F.: A CFD-DEM study of the cluster behavior in riser and downer reactors. Powder Technol. 184, 151--165 (2008)

 Copyright © 2014 SERSC