Numerical investigating flow characteristic of swirling turbulent round jet

B Yuan1,2,3*, S Jia1, L M Pan1,3, Q L Wang1, J X Li1 and J R Li1

1.Key Laboratory of Low-grade Energy Utilization Technologies and Systems, Chongqing University, Ministry of Education of PRC, Chongqing, 400044
2.State Key Laboratory of Coal Mine Disaster Dynamics and Control, Chongqing University, Chongqing, 400044
3.Postdoctoral Research Station of Power Engineering & Engineering Thermophysics, Chongqing University, Chongqing, 400044
*corresponding author: boyuanyuan@cqu.edu.cn

Abstract. Swirling waterjet, using swirling and cutting capacity of high velocity waterjet beam, is a kind of technology widely applied in foundation processing, tunnelling, coalbed methane mining, and etc. It is extremely complicated because the existence of high velocity gradient and swirling, nevertheless, knowing unsteady interior flow characteristic and flow pattern which tremendously affects cutting capacity of waterjet is what we concern about. In this article, computational fluid dynamics is utilized to study flow characteristics of swirling turbulent waterjet, convergent nozzle structure with swirling is chosen as prototype and investigational region is selected as waterjet flow could be fully developed, structured mesh and grid number independent is obtained, Reynolds stress turbulence model is employed and standard wall functions is chosen as the near-wall treatment, pressure boundary conditions are applied on inlet and outlet boundaries, the SIMPLE algorithm is employed to couple the pressure and velocity, and time step size and number is selected reasonably. According to numerical simulation results, statistical analysis method, comparative analysis method and dimensionless analysis method is used to analyse central line velocity attenuation, velocity profile of different axial and radial position, flow pattern and vorticity characteristics, influence of different swirling velocity is also investigated, cutting capacity and influence factor of high pressure swirling water jet is evaluated. It is valuable for application of CFD in swirling waterjet field and helpful for understanding interior filed of swirling waterjet flow, meanwhile, this work could also provide reference for further investigation and application about swirling waterjet.

1. Introduction

High-pressure waterjet technology, as a new internationally developed technology in recent decades, has been comparatively widely used in many fields, such as petroleum exploitation, mechanical processing, coal mining etc. In coal mining, high-pressure waterjet cutting and cooling effect can ensure safety of coal mining process and improve efficiency of coal mining. However, coal and gas outburst is a prominent problem of coal mine in mining process, especially for low permeability coal seam. The main reason causing coal and gas outburst lies in adsorption of coalbed methane (CBM) (gas) in coal seam, then, CBM pre-drainage academic conception proposed by researchers could not only prevent coal and gas outburst, but also collect CBM consider as a utilizing resources. Practical industrial application shows that meshing cutting slot using high-pressure water jet in coal seam pre-drainage CBM thinking proposed by academician Li[1,2], subsequently called hydraulic cutting seam technology,
is an effectively practical way to achieve large-scale CBM resources pre-drainage and also prevent and control gas disasters. High-pressure swirling waterjet is used in hydraulic cutting seam technology to acquire different location slot in coal seam, compared with conventional swirling water jets, it is a kind of jet form that high velocity water sprays out in radial direction, and nozzle swirling along axis direction, as a cutting and breaking medium and a force transmitting medium, flow characteristics of high-pressure swirling water jet is seriously effect on performance of hydraulic slotting in coal seam, therefore, it is necessary to investigating flow characteristic of swirling turbulent round waterjet.

In jet flow, the self-similarity of axisymmetric incompressible isotherm turbulent jet is studied by Wygnanski& Fiedler\cite{3}, results show that axis velocity attenuation and self-similarity characteristic of jet is confirmed; Hussein et al\cite{4} experimental tested characteristic of circular free jet similarity and three order momentum moment under high Reynolds number, results declared that velocity distribution of jet also has self-similarity, and also velocity attenuation of jet axis and conservation of energy is approved. About impinging jet, Lamont&Hunt\cite{5} discussed structural characteristic of turbulent jet under different impact angles, The schlieren images show that shock waves on impact surface is vibrational, and frequency is related to characteristic of nozzle characteristic; Henderson et al\cite{6} measured structure characteristics of supersonic incompletely expansive jet impacting on infinite plate, results demonstrate that stability of jet flow primarily depends on impact strength and velocity gradient in front of impingement; Schwarz &Coasart\cite{7} experimental tested average statistic velocity distribution of impinging jet, results show that dimensionless velocity distribution quasi-curve obeys same distribution, additionally, velocity distribution at the maximum velocity and outer region is different from that of free jet and mixed region; Lanuder&Rodì\cite{8,9} reviewed experimental research on wall jet, self-similar characteristic of velocity distribution using the maximum velocity and velocity half width to be dimensionless is verified. Dejoan&Leschziner\cite{10} using large eddy simulation(LES) to calculate flow characteristics of wall jet, it is shown that LES could effectively simulate flow field two-dimensional wall jet; Ahlman et al\cite{11} using direct numerical simulation(DNS) to simulate scalar mixing characteristics of wall jet, It is found that pressure gradient boundary layer was emerged in inner region, and external region is similar to free plane jet, downstream growth rate is the same as half width growth rate of axis velocity; Shinneeb et al\cite{12,13} investigated jet flow characteristic inferred by offside wall and water surface, results indicated that the center velocity attenuation rate decreased, which is an important effect on jet diffusion rate and horizontal velocity distribution, vertical velocity in downstream region is negative, but horizontal velocity is little affected, there is a large number of vortex structures, level decreasing with downstream but scale increasing, but confined by vertical confinement; Modini et al\cite{14,15} and Wang et al\cite{16} preliminarily discussed swirling jet basic theory and application models; Hiroshi et al\cite{17} using CCD camera experimental studied flow characteristics of swirling jet, results show that flow characteristics is related to nozzle exit velocity, velocity attenuation increases with increasing distance to exit, however, attenuation of velocity is very small and water jet has adequate impact pressure. In this article, numerical simulation method is used to simulate flow characteristics of swirling turbulent round waterjet, reliability and credibility of simulation results is verified with results of particle image velocimetry (PIV) experiment data and reference, characteristics and influence of swirling number is investigated.

2. Numerical approach and verification

2.1. Computational domain and mesh

Depending on practical application, physical simulation model is established(figure 1), confined height $H$, nozzle exit location diameter $D_1$ and flow region $D_2$ is $H/D=10$, $D_1/D=20$, $D_2/D=160$ respectively, nozzle swirl around z axis with a certain swirl number and spray out to a ring confined space, nozzle structure is convergent, convergence angle and diameter $D$ is 140 and 4 mm respectively (figure 2).

Swirling number $S$, which is dimensional number and corresponding with rotating speed of drill pipe, is 0, 0.5, 1.0, 1.5, 2.0, 3.0, 4.0 respectively, calculation formula as follow:
\[ S = \frac{3000v}{\pi U_e} \]  

Where, \( v \), \( U_e \) is respectively nozzle exit circumferential velocity and mean velocity.

Mesh scheme is using unstructured mesh scheme, local refinement of the mesh is employed to improving simulation effective (figure 3), and mesh number is about 12 million.

### 2.2. Numerical method and boundary condition

Turbulent model is choosing RSM turbulent model, the continuity and momentum equation are discreted by finite volume method, pressure discretization scheme is choosing standard and first order upwind is employed in momentum, turbulent kinetic energy and turbulent kinetic energy dissipation rate; The SIMPLE algorithm is choosing to couple the pressure and velocity, and time step size and number is selected reasonably; Pressure boundary conditions is applied on inlet and outlet boundary, Non-slip boundary condition is using in wall and the near-wall treatment is chosen standard wall functions; The sliding meshes model is applied to process rotor-stator interaction.

### 2.3. Verification

Figure 4 is nozzle exit axial average and fluctuating velocity distribution comparison between numerical results and experimental data obtain from Shinneeb[12]. Results shows that axial average velocity distribution is typical “U” distribution, the core velocity is concentrated in center of jet outlet and velocity is almost little attenuation with diameter direction, also, thickness of boundary layer is relatively small; Turbulence in core region is relatively small degree except in boundary region primary causing shear layer. Compared with numerical and experimental results, it is found there is certain differences, numerical velocity distribution in core axis is wider than experimental data, but the maximum velocity is smaller, it is due to larger surface roughness of experimental nozzle, which leads to increase of boundary layer thickness and narrow of core velocity in experimental data. In axial fluctuating velocity distribution, numerical results show that central region is about 5%higher than experiment but lower near boundary layer. These differences maybe subsequently cause dynamic characteristic difference, at nozzle exit, the numerical velocity is basically consistent with experimental. The nozzle outlet velocity profile is basically identical.
Figure 5 and figure 6 shows central axis velocity attenuation and average velocity distribution comparison between numerical results and PIV experimental data in two-side wall confinement. It is shown that axis velocity attenuation curve continuous increases firstly, then increasing rate decreases in a certain distance, in initial phase, experimental attenuation rate is higher than numerical, and the attenuation rate is basically same after attenuation rate reduced, in general, attenuation curve is in good agreement; For average velocity distribution, The experimental dimensionless velocity distribution is “V” type, but not completely symmetrical, nevertheless, numerical data are completely symmetrical. generally, it is found that most of data are basically overlapped.

Figure 4. Nozzle exit axial average and fluctuating velocity profile comparison

Figure 5. Central axis velocity attenuation

Figure 6. average velocity distribution

3. Results and discussion

3.1. Comparison swirling with non-swirling turbulent waterjet

Figure 7 and figure 8 is average velocity and turbulent kinetic energy contours distribution comparison between swirling and non-swirling turbulent waterjet where S=1.0. Results shows that averaged velocity and turbulent kinetic energy distribution of non-swirling turbulent waterjet is symmetrical, while swirling is biased toward one side because of existence of swirling, average velocity and turbulent kinetic energy in core region is affected by swirling and deflection angle becomes greater with downstream from nozzle exit, to average velocity, distribution of back and forward swirling region is different, influence to flow and entrainment on back region is relatively smaller than forward, which is also the reason for biased flow of the subsequent flow, to turbulent kinetic energy, swirling also has a more obvious distribution effect, distribution of the non-swirling is uniform, velocity contours shows "U" type cash band in nozzle and symmetrically distributed in two sides, turbulent kinetic energy gradually increases to the maximum and then gradually decreases with downstream, but for swirling,
"U" type is longer and has a certain bias, turbulent kinetic energy distribution along jet axis is not uniform.

Figure 9 shows central axis velocity attenuation comparison with swirling and non-swirling waterjet where \( S=0.1 \), it illustrates that central axis velocity of swirling and non-swirling also gradually decreases with increasing distance from nozzle exit. At initial stage, energy of swirling is larger than that of non-rotating, which is due to energy carried by swirling nozzle, however, with jet developing downstream, central axis velocity of swirling flow attenuates faster, at distance about \( 14D \), central axis velocity is basically identical, it is because of redundant energy carried by swirling has been consumed in decay process, as developing downstream, central axis velocity of swirling flow attenuates faster, at distance about \( 33D \), the ratio between swirling and non-swirling is about 0.94, that is the ratio of swirling energy attenuation is more about 6%; For linear fitting, it is found that linear coefficient of swirling is 0.059, and non-swirling is 0.053, that is average velocity attenuation rate of swirling is about 11.3% times higher than non-swirling. According to average velocity attenuation characteristic, it is found that swirling increases attenuation rate of central axis velocity, it illustrates that average velocity distribution and turbulent kinetic energy bias of swirling increases attenuation rate of central axis velocity, also declares that swirling increases energy attenuation, strengthens mixing performance and enhances mixing around environment.

3.2. Flow characteristics of swirling waterjet

Figure 10 is averaged velocity distribution contours at different \( z \)-section where \( S=1.0 \), it is found that distribution of velocity contours on different sections is different, however, distribution of velocity contours at the same \( \text{abs}(z/H) \) is basically identical. It shows that velocity distribution of swirling in confined space is symmetrical to \( z=0 \) plane, and also flow characteristic is basically the same because of double-wall confinement; With increasing \( \text{abs}(z/H) \) value, the same with away from jet center, initial averaged velocity distribution is far away from nozzle exit, it indicates that jet is not yet developed into this region, which is basic characteristics of jet flow, also spatial and temporal averaged velocity distribution is decreased. Simultaneously, core velocity is disappeared when \( \text{abs}(z/H) \) is not zero, It indicates that flow characteristic is weakening and far away from the core velocity region, flow energy has been gradually weakened, However, according to contours, color difference decreases relatively less
in downstream, indicates that energy attenuation in longitudinal direction is much slower than flow
downstream attenuation, also that double-wall confinement effect restricts flow characteristics
developing and energy attenuation in longitudinal direction, swirling does little change this confinement.

Figure 10. averaged velocity distribution contours

3.3. Flow characteristics of swirling waterjet influenced by swirling number

Figure 11 is streamline distribution influenced by swirling number at z=0 section. It is found that
streamline distribution in front swirling region is basically identical when swirling number is 0.5, 1, and
1.5, but different in region distribution, to back swirling region, streamline curvature increases gradually,
but bending region decreases. When swirling number is equal to 2.0, there are two huge vortices
appeared in front swirling region, back swirling region also forms streamline bending and curvature
degree is larger than the above swirling number, also bending region is also smaller. When swirling
number is equal to 4.0, distribution is all in confusion, vortex has a tendency to be formed in front
swirling region, it shows that with increasing of swirling number, vortices in front swirling region
influence flow characteristic and result in flow deflection, vortices gradually generate and disappear
leading to pulsation. The larger the swirling number, stronger with deflection and bending extent in back
swirling region, also bending region gradually decreases, local disorder is formed at the maximum
swirling number, which indicates that swirling number affects flow structure and flow characteristic, the
stronger swirling, the greater influence.

Figure 11. Streamline distribution influenced by swirling number at z=0 section

Figure 12. Average velocity contours influenced by swirling number at z=0 section

Figure 12 and figure 13 is average velocity and turbulent kinetic energy contours influenced by
swirling number at z=0 section respectively. It can be seen that average velocity and turbulent kinetic
energy is all affected by increasing swirling number. For average velocity distribution, core length is
slightly increasing with increasing swirling number, but meanwhile deflection angle also increases, the
reason for slightly increasing of core length is that mixing and entrainment are asymmetric due to the
existence of swirling, in back swirling region mixing and entrainment effect slightly decreases, with
increasing swirling number, radial velocity increases causing total energy increases, this increasing
strengthens radial direction movement making mixing and entrainment effect becomes strong, thus
increasing deflection angle. For turbulent kinetic energy contours, with increasing swirling number,
turbulence kinetic energy no longer exists asymmetry along axis, and the larger swirling number is, the more obvious asymmetry is, turbulent kinetic energy distribution is near nozzle exit in back swirling region than front swirling region and becomes more obvious with increasing swirling number, it demonstrates that mixing and entrainment effect in back swirling region decreases.

![Figure 13. Turbulent kinetic energy contours influenced by swirling number at z=0 section](image1)

![Figure 14. Central axis velocity attenuation comparison with swirling number](image2)

Table 1. Central axis velocity attenuation rate comparison with swirling number$^a$

| S  | AR    | C0  | C1  | C2  | C3  | C4  | C5  | C6  |
|----|-------|-----|-----|-----|-----|-----|-----|-----|
| 0  | 0.053 |     |     |     |     |     |     |     |
| 0.5| 0.059 | 11.3| 11.3| 13.2| 13.2| 15.1| 18.9|
| 1.0| 0.059 | 0.060| 0.060| 0.061| 0.063 |

$^a$ AR and RM is attenuation rate and relative magnitude respectively, relative magnitude is calculated by $\text{abs}(C_S - C_0)/C_0$, where $C_S$ is attenuation rate corresponding swirling number, $C_0$ is non-swirling attenuation rate.

Figure 14 and table 1 is central axis velocity attenuation and central axis velocity attenuation rate comparison with swirling number respectively. It is found that dimensionless average central velocity of swirling waterjet is larger than 1.0 in initial stage, and also increase with increasing swirling number, it is shown that with increasing swirling number, energy carried by jet increases gradually, with developing downstream, average velocity begins attenuation; To a certain dimensionless distance, attenuation curve of swirling waterjet intersects with non-rotating curve, and intersection point being away from nozzle exit decreases with increasing swirling number, It shows that flow characteristics is affected by swirling number, energy exchange and mixing with environmental flow is increased , also increases attenuation rate of central axis average velocity. Linear fitting attenuation curve, attenuation rate is obtained, shown in table 1, it can be seen that attenuation rate increases with increasing swirling number, compared with non-swirling attenuation rate, the relative attenuation rate is reaching 18.9% at $S=4.0$, which greatly enhanced waterjet velocity attenuation, therefore, mixing effect of swirling turbulent round waterjet is strengthened by swirling number, but cutting effect is weakened, here it is necessary to explain that capacity of cutting effect is only taken velocity effect into account, impact angle which also influences cutting effect is not involved.

4. Conclusion

Reynolds stress equation model is employed to simulate flow characteristics of swirling turbulent round jet and investigated influence of swirling number, dynamic characteristics and influencing characteristic of swirling number is analyzed, results show that:

(1) According to average velocity attenuation, average velocity distribution and turbulent kinetic energy bias of swirling turbulent round jet increases attenuation rate of central axis velocity, also energy attenuation, strengthens mixing performance and enhances mixing.

(2) Flow velocity turbulent kinetic energy and flow structure is significantly influence by swirling number. The larger swirling number is, averaged velocity distribution biased toward back swirling region level is deeper, turbulent kinetic energy symmetry distribution among jet axis disappears and
becomes more obvious with increasing swirling number, and front swirling region is stronger than back swirling region.

(3) vortex structure appeared in front swirling region causes deflection and oscillation, the stronger vortex structure is, the greater swirling number is, causing significantly increase of streamline bending degree in the back-swirling region. Meanwhile, increasing swirling number leads to increase velocity attenuation rate and strengthen mixing effect, but weaken cutting effect.

Acknowledgement
This work is supported by National Natural Science Foundation of China (Grant No. 51506014), the Fundamental Research Funds for the Central Universities (Grant No. 106112016CDJXY145501), the Chongqing Research Program of Basic Research and Frontier Technology (Grant No. cstc2016jcyjA0486), the China Postdoctoral Science Foundation (Grant No. 2016M592638, 2017T100677), the National Key Basic Research Program of China (Grant No. 2014CB239200) and the Chongqing Postdoctoral Research Program(Grant No. 2016073).

References
[1] Li X H, Lu Y Y and Xiang W Y 2007 Waterjet theory and its application in mining engineering (Chongqing: Chongqing University Press)
[2] Li X H, Lu Y Y, Zhao Y, Kang Y and Zhou D P 2008 J CHINA COAL SOC 33 1386-1390
[3] Wygnanski I and Fiedler H 1969 J FLUID MECH 38 577-612
[4] Hussein H J, Capp S T and George W K 1994 J FLUID MECH 258 31-75
[5] Lamont P J and Hunt B L 1980 J FLUID MECH 100 471-511
[6] Henderson B, Bridges J and Wernet M 2002 8th AIIA/CEAS Aeroacoustics Conf. 17-19 June 2002 Breckenridge, Colorado
[7] Schwarz W H and Coasart W P 1961 J FLUID MECH 10 481-495
[8] Launder B E and Rodi W 1979 PROG AEROSP SCI 19 81-128
[9] Launder B E and Rodi W 1983 ANNU REV FLUID MECH 15 429-459
[10] Dejoan A and Leschziner M A 2005 PHYS FLUIDS 17 025102
[11] Ahlman D, Brethouwer G and Johansson A V 2007 PHYS FLUIDS 19 065102
[12] Shinneeb A M 2006 Confinement effects in shallow water jets (Canada: Ph.D Thesis, University of Saskatchewan)
[13] Shinneeb A M, Bugg J D and Balachander R 2011 J FLUID ENG 133 011203
[14] Modoni G, Croce P and Mongiovi L 2006 GEOTECHNIQUE 56 335–347
[15] Modoni G, Croce P and Mongiovi L 2008 GEOTECHNIQUE 58 533–535
[16] Wang Z F, Shen S L and Xu H S 2012 CHIN J GEOFETCH ENG 34
[17] Yoshida H, Uemura K, Yoshida K and Kataoka I 2007 Proc. of 5th Joint ASME/JSME Fluids Engineering Summer Conf. July 30-August 2 2007 San Diego, California