Numerical simulation of multiphase flow inside hydrocyclone based on CFD

Yuekan Zhang, Peikun Liu*, Linjing Xiao, Xinghua Yang, Junru Yang

College of Mechanical & Electronic Engineering Shandong University of Science and Technology, 266590, Qingdao China

Received 1 October 2014, www.cmnt.lv

Abstract

This paper applied computational fluid dynamics (CFD) method to investigate the internal multiphase flow filed in a 75 mm hydrocyclone. The Reynolds stress model (RSM) and VOF model were employed in the numerical simulation. This study discussed the velocity and pressure distribution in the hydrocyclone, and analysed the formation and development mechanism of air core. The numerical simulation results showed that the flow field was very unstable in the region of the air core. The axial velocity gradient reached its maxima, and the turbulent fluctuation was strongest in the simulation region. This study provided theoretical basis on further research of the air core effect on separation efficiency and pressure drop.

Keywords: numerical simulation, multiphase flow, hydrocyclone, CFD

1 Introduction

Hydrocyclone is a typical device that is used to separate non-uniform multiphase fluids. Owing to its advantages of small space requirement, simple structure, high separation efficiency, large separation capacity, low maintenance costs, and no moving parts, it has been widely used in various industries such as chemical engineering, oil/gas, and ore dressing. Although the structure of hydrocyclone is simple, the inner flow field is very complex and the flow characteristics are complicated. High speed rotating turbulent flow, interaction between multiphase flows and interruption of the air core increase the challenge to the study of internal flow field in hydrocyclone [1-3]. For many years researches on hydrocyclone were typically carried out in experimental ways. For example, Kesall [4] applied light-speed microscopic measurement system, Knowles et al. [5] used imaging velocimetry system, and Hsieh et al. [6] employed a LDV to study the internal velocity distribution of hydrocyclone, in which small amount of tracing particles were added to the water. However, these methods require delicate experimental conditions, high cost, long working period, and have disadvantage of difficult measurement and non-universality. With the development of computation fluid dynamics (CFD) and continuous improvement of turbulent models, numerical simulation becomes research hot points in hydrocyclone studies. Through simulation, the internal flow field and velocity distribution can be clearly detailed, which provides important guide to structural design and optimization of hydrocyclones. This paper applied a CFD software, FLUENT, to simulate the internal two-phase flow field in a hydrocyclone. This study correctly reflected the internal flow field in the hydrocyclone, obtained the velocity and pressure distribution, and especially focused on the formation and development mechanism of the air core.

2 Construction of the model

Structural model This paper aimed at the standard hydrocyclone that was proposed by Hsieh, [6] where the structural dimension is shown in Figure 1.

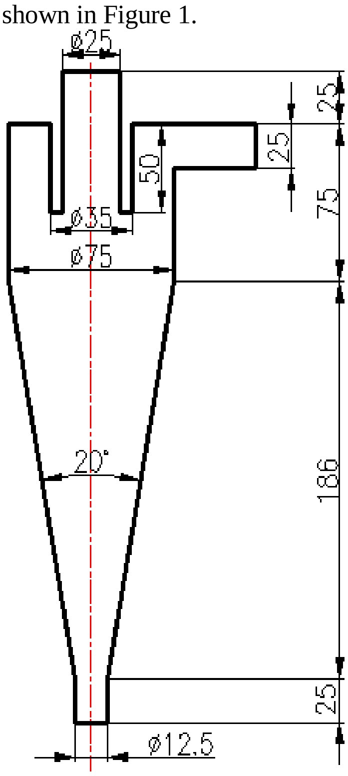

FIGURE 1 Geometry of the hydrocyclone

The diameter of cylindrical section Φ is 75mm, and the length is 75mm. The cone section is 186mm long, and the length of the underflow section is 25mm. The inner depth of the overflow section is 50mm. The overflow tube thickness

**Corresponding author* e-mail: lpk710128@163.com

is 5mm and the diameter of the overflow and underflow tubes is 25mm and 12.5mm, respectively. The inlet section area is 25×12.5mm, and the cone angle is 20°. The mesh was generated through ICEM with structural hexahedral mesh. The mesh in the central axis and wall area was refined, which is shown in Figure 2.

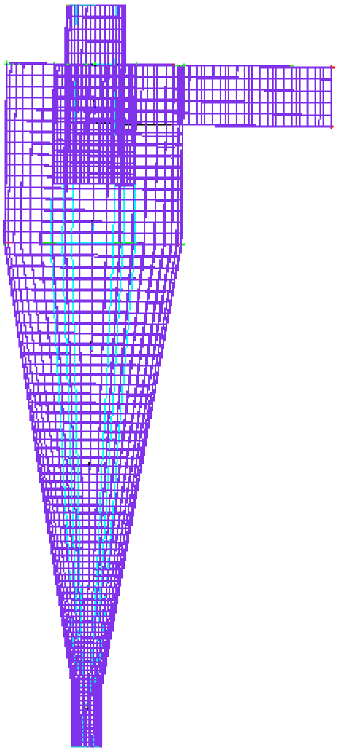

FIGURE 2 The mesh of the hydrocyclone

Determination of the turbulent and multiphase flow models Due to the complexity of turbulent flow, its generation mechanism has not been fully understood. Bhaskar et al. [7] used experimental studies to compare k-ε turbulent model, k-εRNG turbulent model and RSM model on the numerical simulation of hydrocyclone flow. The results indicated that the RSM model could provide the smallest error compared to the experimental data, because RSM model completely considers the rapid change of streamline curve, rotation and tension, which can well predict anisotropic turbulence. Therefore, it is very suitable for the study in hydrocyclone flow where the flow is highly rotational. Considering the flow in the hydrocyclone is two phase or multiphase flow, this paper applied volume of fluid (VOF) model to study liquid-air two phase flow. This study solved a set of momentum equation and analysed the VOF of two phase flow through the computational domain, which can well simulate the un-dissolved fluid distribution of air and water in the hydrocyclone.

Boundary condition and controlling parameters Boundary condition is the key to determine the exact solution of the governing equation. The governing equation and boundary condition constitute the complete mathematical description of the flow process of a physical variable. FLUENT provides various boundary conditions. In this study, the boundary condition was set as follows:

Inlet condition: the primary phase was set to be water. The temperature is constant, density of water is 998.2kg/m^3, viscosity is 0.001Pa·s, volume fraction is 1. The secondary phase was set to be air, the temperature is also constant, density is 1.225kg/m3, viscosity is 0.00001 Pa·s and volume fraction is 0. The velocity of water at inlet was 2.28m/s and the turbulent characteristics were determined based on the combination effect of hydraulic diameter and turbulent level.

Pressure outlet: Both overflow and the underflow exits were set exposed in the air, where the relative pressure was zero. The backflow coefficient of air was set to be 1. In this regard, the backflow of fluid would be air, if negative pressure presented at the flow outlet.

Wall condition: The wall condition was treated as non-slip boundary condition, where the boundary layer turbulence was determined according to the standard wall function.

3 Simulation results and discussion

The residual was set to be 10^{-5} and the time step was 1×10^{-4}s. The velocity field, pressure field, and the formation and development of the air core were determined through iteration. The results and discussion are presented as follows.

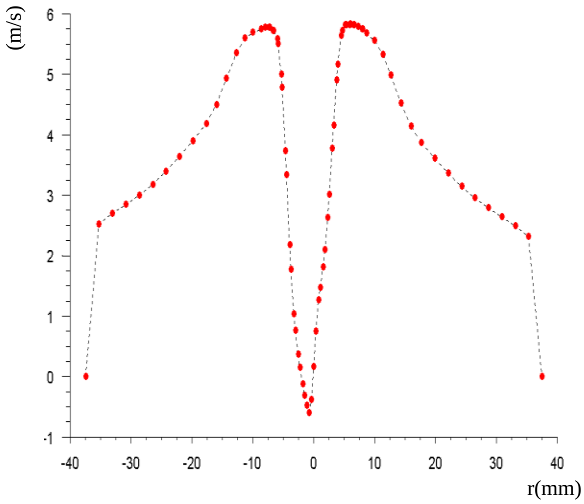

FIGURE 3 The tangential velocity distribution

Tangential velocity distribution in the hydrocyclone, tangential velocity has great influence on the separation efficiency. Large tangential velocity has great centrifugal force, and therefore high separation efficiency. From the tangential velocity distribution (Figure 3), tangential velocity varied clearly along the axis, where the minimum tangential velocity was at the wall. With the gradual decrease of radius, the value of tangential velocity increased continuously. After reaching the maximum value at the air core edge, the tangential velocity tended to decrease gradually. The distribution of the tangential velocity had the characteristics of combined vortex. An obvious transition area existed between forced vortex and free vortex, which agreed well with the research results of Kelsall [4] in 1952.

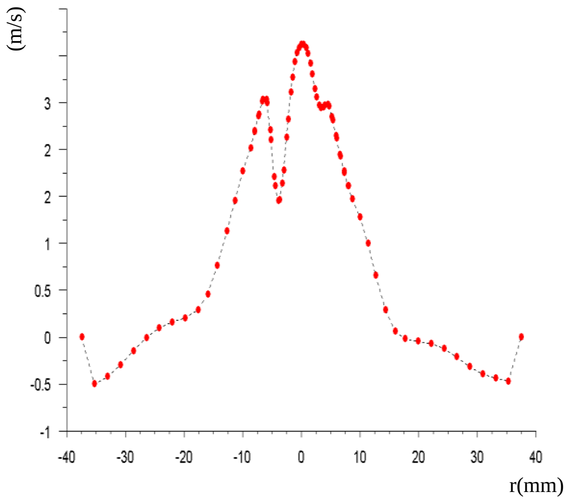

FIGURE 4 The distribution of axial velocity

Axial velocity distribution: during the separation process of the hydrocyclone, axial velocity directly affects the retention time of the fluid in the flow field, and determines the distribution relationship of fluid between overflow and underflow. Hsieh [6], Kelsall [4], and Bradley et al [2] conducted detailed experimental studies of axial velocity in hydrocyclone. Using experimental data, Xu [8] applied mathematical regression method to fit the mathematical equations for axial velocity distribution of fluids in the area under the overflow tube of hydrocyclone. The distribution of axial velocity in this study is shown in Figure 4. The direction of flow in the tube was downward close to the wall, towards the underflow exit. Towards the axial direction in the tube, the axial velocity decreased with the decrease of radius. The axial velocity was zero at the location of approximately half radius. After this zero point, the axial velocity changed to upward direction, i.e., the flow was towards the overflow exit, and increased with the decrease of radius. There was always a point with zero axial velocity at different cross-sections. Therefore, a contour surface shaped like a cone can be named locus of zero vertical velocity (LZVV), shown in Figure 5. This LZVV divided the axial velocity of fluids in the hydrocyclone into two parts: The flow direction of fluids outside of the zero axial velocity envelope plane was downward, forming outer helical flow; In contrast, the flow direction of fluids inside of the zero axial velocity envelope plane was upward, forming internal helical flow. Figure 6 gives axial velocity gradient of hydrocyclone flow. It can be seen that the gradient close to the wall was smallest. With the decrease of radius, in the area of air core, the gradient increased rapidly until reaching a certain value and then decreased fast, forming a hump-like distribution. It indicates that turbulence in the centre area of the hydrocyclone was very intense, and the flow field was extremely unstable. However, the turbulent level was weakest in the area of hydrocyclone wall, where the flow field was relatively mild.

FIGURE 5 The locus of zero vertical velocity

Pressure distribution: The pressure distribution in hydrocyclone is very important to separation efficiency and separation size, which is the primary basis for calculation of productivity and energy loss. Through the two-phase numerical simulation, the pressure distribution in the hydrocyclone is obtained and shown in Figure 7. It can be seen that the static pressure in the hydrocyclone gradually decreased from the wall towards the axis. The static pressure of the fluids was zero at the location of the air core interface. Afterwards, the negative pressure became stronger as closer to the axis, and decreased to the minimum value at the underflow and overflow exits.

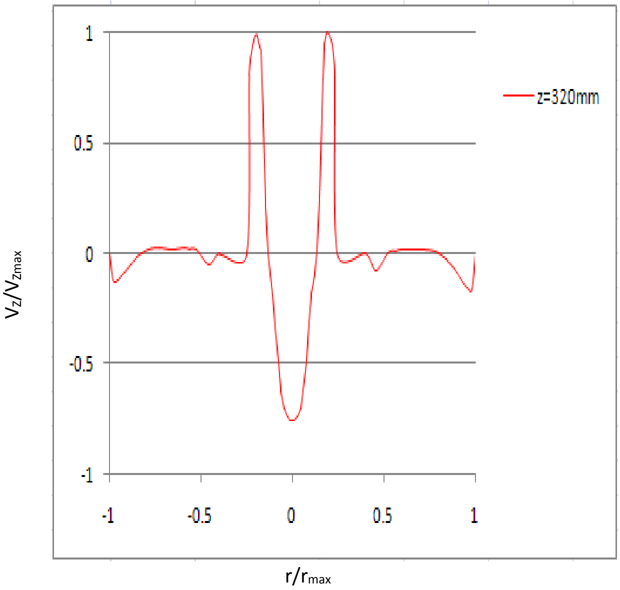

FIGURE 6 The axial velocity gradient of the hydrocyclone

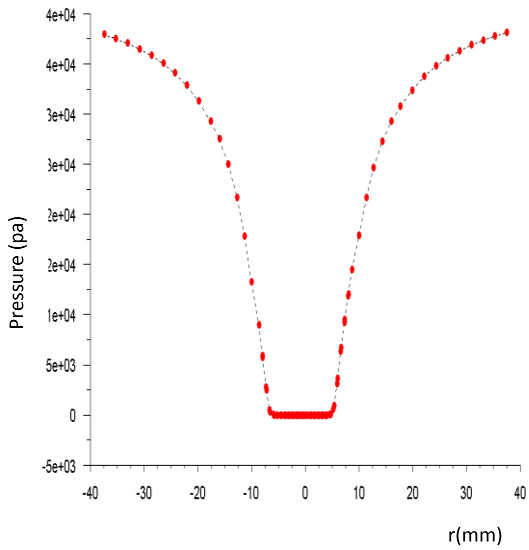

FIGURE 7 The pressure distribution of the hydrocyclone

Formation and development of the air core Figure 8 shows the formation and development of the air core in the hydrocyclone. In the figure, blue color represents air, whereas red part is water. At the initial time, the hydrocyclone is filled with air. With the evolution of the flow field, the volume of air in the hydrocyclone decreased gradually, and the ratio of water phase volume to total volume increased. When the time is about 1.1s, water occupied the whole internal area of the hydrocyclone. With the increase of time, air core initially formed under the overflow tube, and then in the area of underflow exit. The two air core grew and moved towards each other until they merged to a single air core. It is evident from this numerical simulation that, because the underflow and overflow tubes were directly connected to air, the fluids entering into the hydrocyclone rotated strongly as spiral vortices. When the tangential velocity reached a certain value, the negative axial pressure formed, and air entered into hydrocyclone. Under the effect of this negative pressure, the air core developed and reached the stable condition. The size and shape of the air core changed with the development of the flow field. Even when the flow condition reached stable, the sizes of the air core at different axial locations were not the same. The axial change of diameter of the air core is shown in Figure 9. The maximum diameter of the air core located at the bottom of the overflow tube, which is 15.57 mm. The diameter of the air core decreased to 9.91 mm with the decreasing height of the hydrocyclone, which was induced by structural change of bottom section of the overflow tube. In the cone section, the diameter of the air core varied irregularly at different locations in the hydrocyclone. The characteristics of the air core in the underflow tube were similar to that in overflow tube. The diameter at bottom of the underflow tube is maximum, i.e., 9.8 mm. The diameter changed suddenly to 8.91 mm at the interface between underflow tube and the cone section.

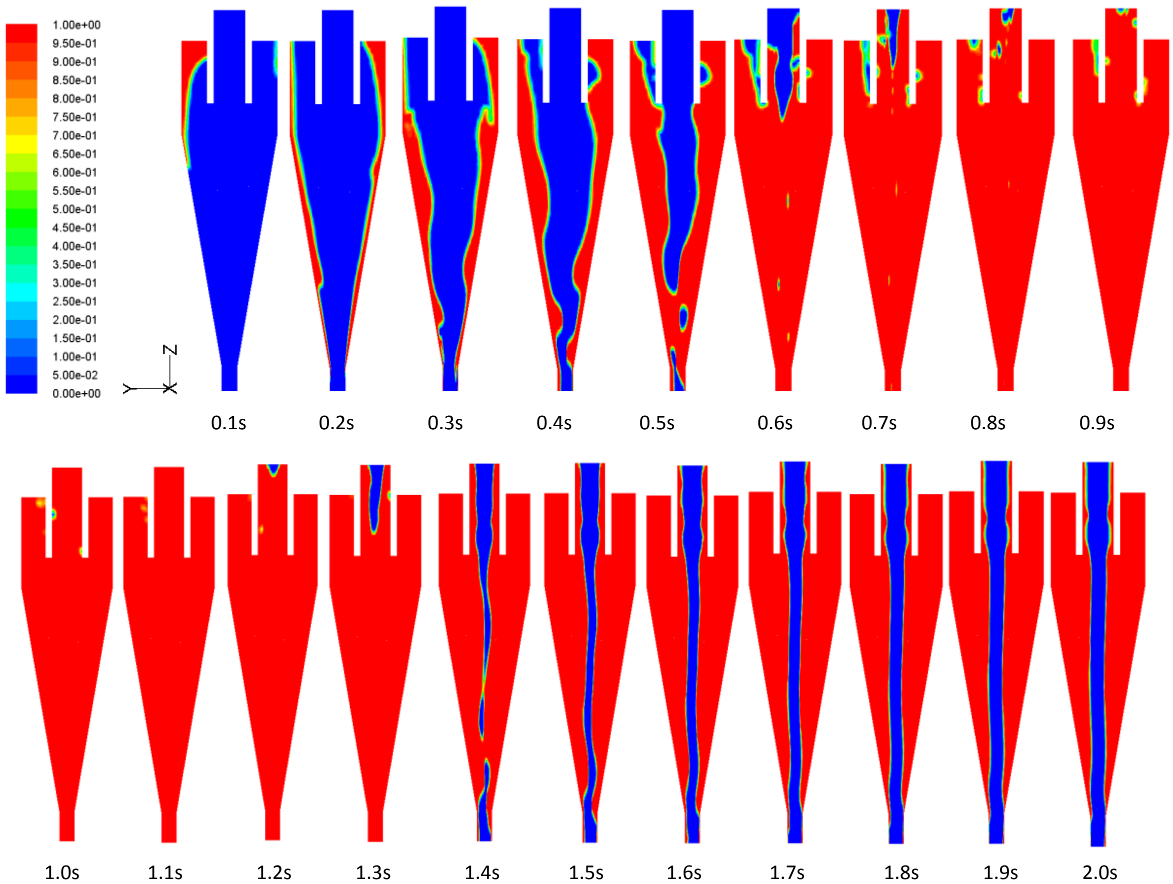

FIGURE 8 The development of the air core in the hydrocyclone obtained from CFD simulations

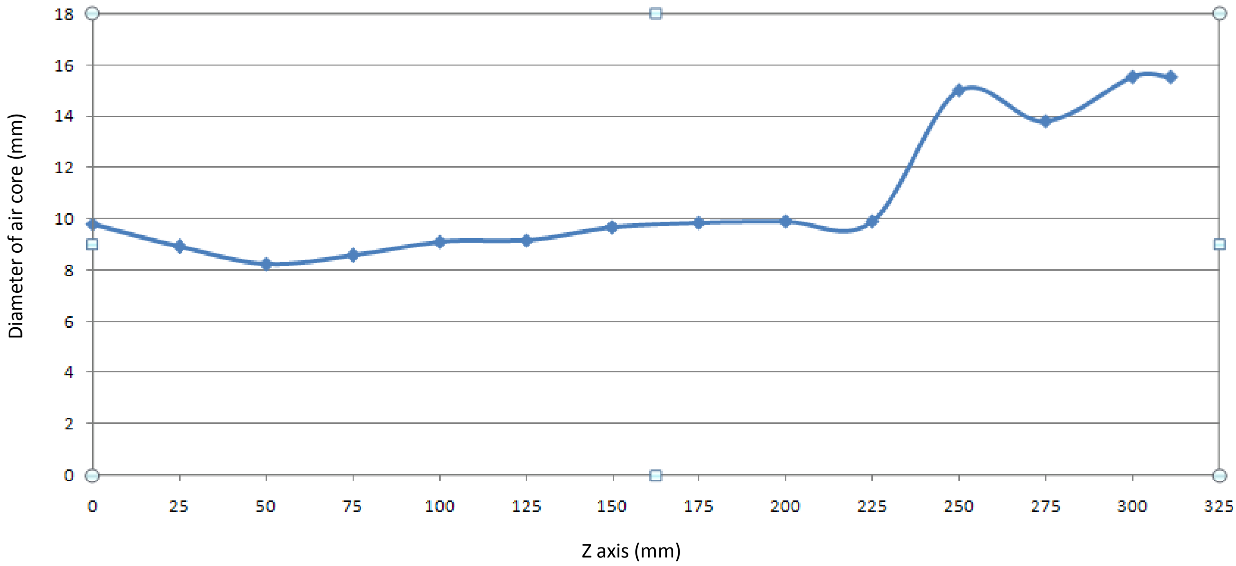

FIGURE 9 The axial change of diameter of the air core in the hydrocyclone

4 Conclusions

This paper used FLUENT and applied RSM and VOF models to simulate the two phase flow field in a hydrocyclone. The results of this study were consistent with the previous theoretical and experimental studies. The results explored the internal flow field in the hydrocyclone. This study emphasized the formation and development of the air core. The numerical results showed that the flow field was highly unstable in the area of the air core, with the largest axial velocity gradient and strongest turbulent fluctuation. This study also provides the theory basis of further investigation of air core effect on the separation efficiency and pressure drop.

Acknowledges

This work is supported by the National Natural Science Foundation of China (No.21276145) and guiding program for scientific and technological research of China National Coal Association (No. MTKJ2013-365).

References

[1] Wills B A, Napier-Munn T 2006 Will's Mineral Processing Technology *7th Oxford Butterworth-Heinemann*

[2] Bradley D 1965 The Hydrocyclone *London: Pergamon Press*

[3] Nowakowski A F, Cullivan J C, Williams R A, Dyakowski T 2004 Application of CFD to modelling of the flow in hydrocyclones. Is this a realizable option or still a research challenge? *Minerals Engineering* **17**(5) 661-9

[4] Kesall D F 1952 A study of the motion of solid particles in a hydraulic cyclone *Transactions of the Institute of Chemica Engineers* **30** 87-104

[5] Knowles S R, Woods D R, Feuerstein I A 1973 The velocity distribution within a hydrocyclone operating without an air core *The Canadian Journal of Chemical Engineering* 1973 **51**(3) 263-71

[6] Hsieh K T 1988 A phenomenological model of the hydrocyclone *Thesis Salt Lake City* University of Utah

[7] Bhaskar K U, Murthy Y R, Raju M R, Tiwari S, Srivastava J K, Ramakrishnan N 2007 CFD simulation and experimental validation studies on hydrocyclone *Minerals Engineering* **20**(1) 60-71

[8] Xu J, Luo Q 1988 Flow fields theory of the hydrocyclone *Beijing China Science Press (in Chinese)*

Authors

Yuekan Zhang, born in March, 1970, Xinji, Hebei Province, China

Current position, grades: instructor at the College of Mechanical & Electronic Engineering Shandong University of Science and Technology. PhD degree student in mineral process engineering at the Shandong University of Science and Technology, China.
University studies: MS degree in mechanical design and theory at Shandong University of Science and Technology, China in 2007.
Scientific interests: computational fluid dynamics, solid-liquid separation technology and equipment.
Publications: 25 papers.

Peikan Liu, born in January, 1971, Wenshang, Shandong Province, China

Current position, grades: professor at Shandong University of Science and Technology.
University studies: Doctor's degree in mineral processing at Wuhan University of Science and Technology China in 2008.
Scientific interests: computational fluid dynamics, solid-liquid separation technology and equipment.
Publications: 30 papers.

Linjing Xiao, born in January, 1966, Yishui, Shandong Province, China

Current position, grades: professor at the College of Mechanical & Electronic Engineering at Shandong University of Science and Technology.
University studies: PhD degree at Beijing University of Science and Technology, China, in 2001.
Scientific interests: virtual prototype and CAD/CAE/CAE.
Publications: 55 papers.

Xinghua Yang, born in October, 1978, Luoshan, Henan Province, China

Current position, grades: instructor at the College of Mechanical & Electronic Engineering Shandong University of Science and Technology.
University studies: MS degree at Shandong University, China, in 2001.
Scientific interests: computational fluid dynamics, solid-liquid separation technology and equipment.
Publications: 28 papers.

Junru Yang, born in October, 1969, Leting, Hebei Province, China

Current position, grades: associate professor at the College of Mechanical & Electronic Engineering Shandong University of Science and Technology.
University studies: PhD degree at Shandong University, China, in 2006.
Scientific interests: cladding part design and CAD/CAE
Publications: 40 papers.