

Numerical simulation of aerodynamic characteristics of windmill based on computer graphics display technology

Jingbin Hao*, Haifeng Yang, Zhongkai Li

College of Mechanical and Electrical Engineering, China University of Mining and Technology, Xuzhou, China 221116

Received 1 May 2014, www.cmnt.lv

Abstract

This paper proposed research on wind computation fluid dynamics(CFD) analysis and visualization in scientific computing facing to OSG in order to obtain numerical result of aerodynamic characteristics stimulated by windmill and to display vivid and direct 3-D effect for easy observation experiment. This paper applied CFX module in ANSYS Workbench software to conduct CFD analysis and calculation of windmill through computer graphic display technology based on CFD principle and realize numerical stimulation of aerodynamic characteristics of windmill. Then the research on visualization in scientific computing of aerodynamic characteristics of windmill was realized combining with OSG software that face to virtual display technology

Keywords: computation fluid dynamics, computer graphics display technology, numerical stimulation, 3-D geometric model

1 Introduction

Various countries in the world are searching for new energy shortcut for the shortage of world energy [1]. Therefore, wind energy becomes an area, to which many countries pay high attention. For large volume and super high large wind driven generator, wind load is a quite important control load. However, problems such as wind pressure distribution and smooth interference usually cannot solved by normative solution pattern when important parameter of wind load is mentioned. Traditional action is to put entity or integration model into wind tunnel to conduct analysis and stimulation experiment of relative data so that obtain relative wind load experimental data result [2], which is used for design and improvement data of relative entity model. For instance, when airplane and car is done improved design, we usually extract relative experimental data in wind tunnel experiment [3]. However, wind tunnel has drawbacks of insufficient scale effect, high cost, long period and bad flexibility.

Scientific visualization has become the important research direction of current computation graphics [4]. People use traditional original visualization technology method to count or draw chart on scientific analysis data for observation experiment [5]. We aims to display 3-D flow field result figure generated by numerical simulation of aerodynamics characteristics of windmill in vivid and direct effect, make users observe the process when wind blowing over windmill more directly, increase perceptibility of windmill flow field and improve scientific research level of windmill. Therefore, we introduce "numerical wind tunnel" by combination of software and hardware for conducting numerical

simulation analysis technology of aerodynamics characteristics of relative model [6]. Car virtual wind tunnel research of Jili University and virtual wind tunnel research on aircraft of Beijing University of Aeronautics and Astronautics [7] have problem of few application cases and few departments that can provide 3-D data. Therefore, this paper explored and studied how to realize application of visualization in scientific computing more deeply.

2 Overview of computation fluid dynamics

Computation fluid dynamics (CFD) refers to systematic analysis and research on physical phenomenon such as heat conduction and fluid flow by computer numerical analysis and calculation and graphics display technology [8]. CFD is to conduct numerical simulation on fluid flow under the control of three flow basic equation [9]. Fundamental physical quantity in various position within fluid field and their basic situation of changing over time can be derived from these numerical simulation. CFD can display simulation result on computer screen by specific calculation means. In addition, it can also observe detailed effect in flow field to display flow scene directly and vividly.

Procedures of numerical simulation on fluid flow characteristics by CFD are as follows. First, a mathematical model that reflects problem nature should be set up. That is, to build up integro-differential equation that reflects the relationship between quantities of problem with definite condition and three conservation equation with their definite condition which is easy for calculation simulation solution. Then numerical discrete optimization method should be constructed as the core of

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

CFD, which is easy for research and analysis on CFD. Moreover, compilation and calculation are carried out. It includes division of computer network, import of initial conditions, boundary conditions, setting of control parameter, etc. At last, the result is conducted refine processing to obtain a high quality of display result.

The practical solution process of CFD is very complicated. The application of CFD is also complex. Therefore, many users select CFD commercial software. The structure of this commercial CFD software mainly includes pre-processing, solution and post processing.

3 Pre-processing and data conversion of 3-d geometric model of windmill

3.1 PREPROCESSING OF 3-D GEOMETRIC MODEL OF WINDMILL

Wind turbine blade and overall stand is selected to analyse and calculate and conduct numerical simulation research of aerodynamic characteristics of windmill.

3.1.1 Construction of Windmill 3-D Model

Windmill whose rated power is 1.5 megawatt designed by some enterprise was selected. And UG software produced by Siemens PLM Software Corporation was used for windmill 3-D geometric modelling. Fig.1 is UG windmill 3-D geometric model. Tower in tapered tubular steel structure was made of low-alloy high tensile structural steel Q 345. Hub centre is of 70 m high, 11 m/s rated wind speed, 3.5 m/s cut-in wind speed, 25 m/s cut-out wind speed and 3847 m² swept area. CFX application module in ANSYS-Workbench was selected to do one way fluid-structure interaction analysis to realize calculation process of numerical simulation of aerodynamic characteristics of windmill because UG software itself is hard to realize that process.

3.1.2 Realization of UG and ANSYS Software Data Interface

UG is a software system of CAD and CAM that can realize interaction function. After 3-D model design software UG was used for constructing windmill 3-D geometric model, is was done data conversion and then applied into ANSYS-Workbench software to realize numerical simulation of aerodynamic characteristics of windmill. And IGES file format which is the sharing common format of UG and ANSYS-Workbench. Their characteristic was applied to read complete data from different platform to realize standard of data interchange between CAD and CAM system [10]. Thus, UG software could be used for deriving 3-D geometric model of windmill. Then it was derived into ANSYS-Workbench to conduct windmill 3-D geometric reconstruction to achieve numerical simulation of aerodynamic characteristics of windmill.



FIGURE 1 3-D geometric model of UG windmill

3.2 DATA CONVERSION OF WINDMILL 3-D GEOMETRIC MODEL FROM UG TO ANSYS

We have pre-processed windmill geometric model and realize interface function of data conversion of UG and ANSYS. Therefore, we can directly do data conversion on windmill 3-D geometric model into IGES file format and then import it into ANSYS Workbench software to do research on numerical simulation analysis of aerodynamic characteristics of windmill.

IGES file derived from windmill 3-D geometric model should be prepared before analysis. We selected Start Menu, through the path "ALL Programme, ANSYS14.0 > Workbench", start ANSYS Workbench module, open "fengjizhengti, igs" file in window of "File > Import" path and select Geometry to conduct "Update" geometric model load.

4 Application of CFD module from ANSYS software in numerical simulation of aerodynamic characteristics

Preparation of geometric file was basically fulfilled through the previous work. Next, CFX module in ANSYS software was applied to do numerical simulation of aerodynamic characteristics of windmill. CFD commercial software, which has regularity and systematicness was needed to solve the problem because CFD calculation involves allocation of computer software and hardware and the complexity of problem. CFX was constantly integrated the most advanced global physical geometrical model and simulation solution algorithm for ensuring the accuracy and stability of calculation result. Meanwhile, finite volume method based on finite element was also used for ensuring basic conservation characteristics and accuracy of numerical value [11].

4.1 REALIZATION OF PREPROCESSING OF WINDMILL 3-D GEOMETRIC MODEL IN CFX MODULE OF ANSYS SOFTWARE

Pre-processing of 3-D geometric model mainly involves functional operation in DesignModeler on the model that has been imported to realize modelling process of rotating field and external flow field.

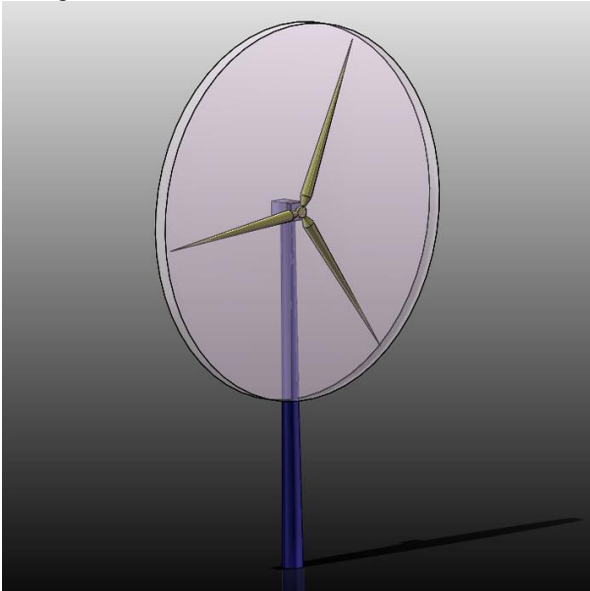


FIGURE 2 windmill model of cylindrical rotating field

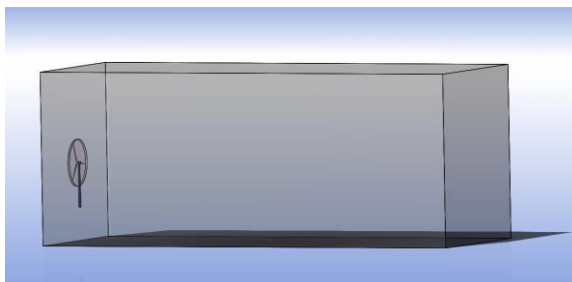


FIGURE 3 square external flow field of windmill

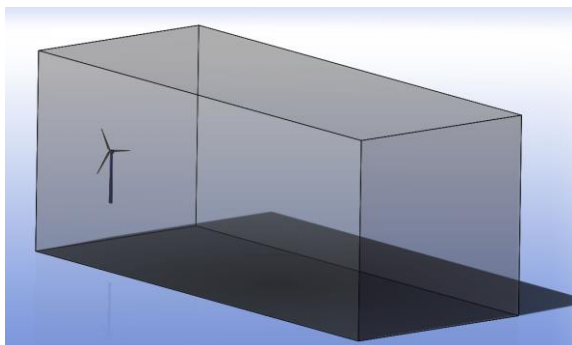


FIGURE 4 square external flow field of windmill after Boolean calculation

First, we double clicked GeometryA2 module to conduct windmill 3-D geometric model load to enter DesignModeler analysis module and selected Meter unit system in “unit” dialog box. Second, we constructed cylindrical rotating field of windmill 3-D geometric model through path “Create > Primitives > Cylinder”. Fig.2 is windmill model of cylindrical rotating field. Then

blade model was removed. We did Boolean calculation of windmill 3-D geometric model through path “Create > Primitives > Box”. We selected “Yes” order in Preserve Tool Bodies for reservation of blade model. At last, after rotating field setting, square external flow field was constructed through path “Create > Primitives > Box”. Then Boolean calculation was done again. Fig.3 is square external flow field. Fig.4 is square external field after Boolean calculation.

4.2 FLOW FIELD MESH DIVISION OF 3-D GEOMETRIC MODEL OF WINDMILL

After pre-processing of 3-D geometric model of windmill by ANSYS Workbench, we used Mesh module to divide 3-D geometric model mesh of windmill to generate the needed calculation mesh.

First, we constructed Mesh module, dragged GeometryA2 into MeshB2 and imported pre-processed windmill 3-D geometric model into Mesh module. Then we expanded path “Project > Model > Geometry” to observe some Solid body. Windmill model was suppressed and only the flow field was remained. Solid body would show out X mark after suppression. Then we revised Solver Preference value as CFX through path “Project > Model (B3) > Mesh”. CFX-meshing under Mesh was right clicked. At last, we realized the setting of boundary condition by setting entrance of external flow field, exit of reserve side of windmill, ground and part surface of windmill stand. Meanwhile, some specific network properties were set. At last, mesh model of windmill flow field is generated as shown in Fig. 5.

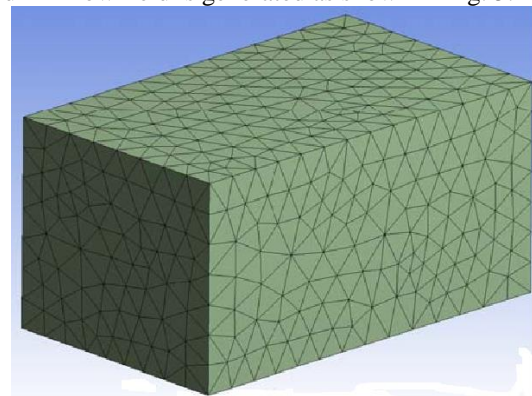


FIGURE 5 Mesh model of windmill flow field

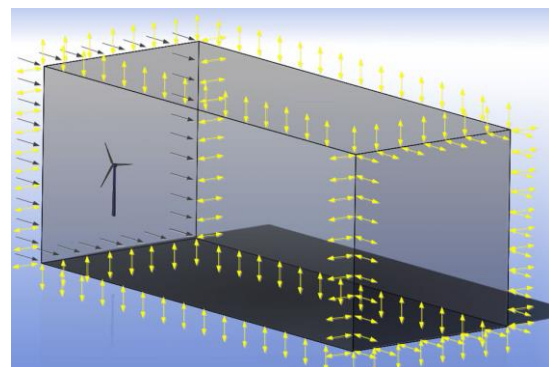


FIGURE 6 load figure of rotating field and external flow field

4.3 SETTING AND SOLUTION OF FLUID ANALYSIS OF WINDMILL 3-D GEOMETRIC MODEL

After division of windmill 3-D geometric model mesh, we entered into fluid analysis setting including designing boundary condition and properties of solver.

First, we set analysis category and imported divided Mesh file into CFX-Pre for fluid calculation. Then CFX module was generated automatically and connected. Then rotating field and external fluid field were constructed. At last, the relative boundary parameter condition was set. Fig. 6 is the load figure of rotating field and external fluid flow field after setting of relative parameters.

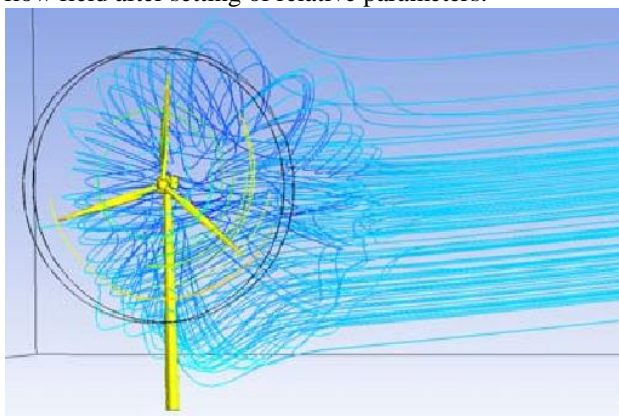


FIGURE 7 windmill 3-D flow field

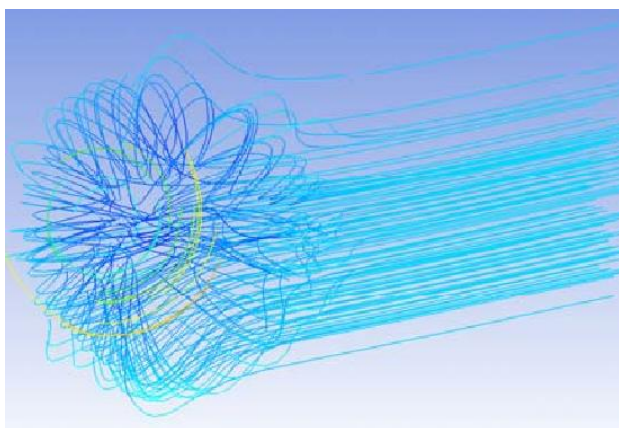


FIGURE 8 flow line of 3-D flow field removing windmill

We only need to do simple solver setting after calculation of fluid and then solution result could be achieved. Fig. 7 is windmill 3-D flow line after setting of solver.

References

- [1] Fengyi C, Shaohua G 2012 Draught fan industry market analysis and prediction *General Machinery* 2 22-26
- [2] Tianshu Z 2007 Research on numerical wind tunnel method of shape factor confirmation of wind load. *Tianjin University*
- [3] Xiaobi S 2010 Aerodynamic computation on wind pressure distribution of ultra-large cooling tower Guangdong Electronic Design Institute *Journal of Wuhan University* 8 270-274
- [4] Huiyan J, Ning L 2007 3-D reconstructed system design based on advanced ray casting method Collected Papers of 2nd 3-D Image Technology and Application (International) Seminar 9 18-20
- [5] Huisen Z 2006 3-D visualization platform design and realization facing to tourist resource management *Shanxi Normal University* 5 3-4
- [6] Tianqin G 2009 Low speed wind tunnel boundary layer control numerical simulation research *Huazhong University of Science and Technology* 5 1-3

Through application of CFX module in ANSYS Workbench software, 3-D flow field figure removing windmill can also be independently produced. Fig. 8 is flow line of 3-D flow field removing windmill.

5 Conclusion

This paper mainly illustrated application and basic knowledge of common commercial software in CFD in order to study windmill CFD. Based on it, we pre-processed windmill geometric model and realized interface function of data conversion between UG and ANSYS. Windmill 3-D geometric model constructed in UG was directly converted into IGES format and then imported into ANSYS Workbench software for study of numerical simulation of aerodynamic characteristics of windmill. IGES file derived from windmill 3-D geometric model should be prepared before analysis. And CFD module in ANSYS software was applied for numerical simulation of aerodynamic characteristics of windmill. CFX module can ensure accuracy and stability of calculation result as well as basic conservation characteristics and accuracy of numerical value. Pre-processing of 3-D geometric model mainly involved functional operation in DesignModeler on model that have been imported to realize the modelling process of rotating field and external flow field. Through pre-processing of windmill 3-D geometric model by ANSYS Workbench, we used Mesh module to divide the mesh of windmill 3-D geometric model to generate the needed calculation mesh. After division of windmill 3-D geometric model mesh, we entered into fluid analysis setting including designing boundary condition and properties of solver. We only need to do simple solver setting after calculation of fluid and then solution result could be achieved. At last, the numerical simulation of aerodynamic characteristics of windmill was achieved.

Acknowledgment

This work was supported by National Natural Science Foundation of China (51305443), Natural Science Foundation of Jiangsu Province (bk20130184), Fundamental Research Funds for the Central Universities (2012QNA27), and National High Technology Research and Development Program of China (863 Program) (2012AA062100).

- [7] Tiezhong L, Qiang L, Zhoufu L 2009 Virtual aircraft wind tunnel research and realization Beijing University of Aeronautics and Astronautics. *Journal of System Simulation* 14 1-3
- [8] Jian L 2002 3-D visualization processing of engine crankshaft element calculation based on OPENGL technology *Shanghai Jiaotong University* 2 1-10
- [9] Tuncer C, Jian P S, Fassi K, Eric L 2009 Computational fluid dynamics for engineers *Beijing: Tsinghua University Press* 4 31-35
- [10] Jining S 2010 Basic application course of ANSYS CFX convection heat transfer numerical simulation. *National Defence Industry Press*.
- [11] He Y, Xiang J 2003 Computer graphics principle, method and application. *Higher Education Press*.

Authors	
	<p>Jingbin Hao, born in June, 1982, Jiangsu Province of China</p> <p>Current position, grades: lecturer University studies: Ph. D degree was earned in major of Mechanical Manufacture and Automation, University of Mining and Technology in 2011. Scientific interest: laser rapid prototyping, machine vision and laser imaging.</p>
	<p>Haifeng Yang, born in May, 1981, Jiangsu Province of Chin</p> <p>Current position, grades: Associate Professor University studies: Ph. D degree was earned in major of Materials Science, Jiangsu University in 2009. Scientific interest: Laser nano-manufacturing, Unconventional Machining Technology.</p>
	<p>Zhongkai Li, born in February, 1980, Shandong Province of China</p> <p>Current position, grades: Associate Professor University studies: Ph. D degree was earned in major of Mechanical Design and Theory, Zhejiang University in 2009. Scientific interest: Product family design, Enterprise informatization</p>