2010 3rd International Conference on Computer and Electrical Engineering (ICCEE 2010)
IPCSIT vol. 53 (2012) © (2012) IACSIT Press, Singapore
DOI: 10.7763/IPCSIT.2012.V53.No.2.73

Application of Parallel Compution on Numerical Simulation for the Fluid Flow Field of Fan

Junjie Zhou+, Xiaoqian Li, Dingbiao Wang, Bo Liu and Jinhui An

School of Chemical Engineering and Energy, Zhengzhou University

Zhengzhou, China

Abstract—In this paper, Numerical simulation and analysis was conducted on the flow field of axial fan by FLUENT software and the solved speeds of serial and parallel solver has been compared, and the effect of different number of nodes on the speed of parallel computation was discussed. The results showed that the convergence time used with the parallel solver is reduced by nearly 40% than that of the serial solver; with the increase of the number of nodes in parallel computation, the time required for convergence first decreased and then increased. The convergence time is the least when it is 6 nodes. On this basis, three-dimensional time average N-S equation was solved by the parallel algorithm, and the flow field of axial fan was simulated. The results show that when air flows through the impeller, vortex flow is generated at the rear of wheel, and the flow resistance increase; The "C" type distribution of pressure gradient is formed at leaf blade surface which can better improve the air distribution in the channel.

Keywords-axial fan;numerical simulation;FLUENT;parallel computation

1. Introduction

Turbomachinery, one of the most important power installations in contemporary society, plays an important role in all sectors of the national economy. The total electrical capacity of motor-driven equipment such as pumps, fans achieves 150 million kw, which accounts for about 35%of electric power generation of the country[1],so increasing the operating efficiency of turbomachinery can make full use of the limited energy and improve economic efficiency; the improvement of design standards and performances of all kinds of turbomachinery depends largely on the understanding of physical nature of the real flow phenomenon[2-3].In order to improve the design quality of the fan, and to design effective fluid device faster and better, we must understand the inside actual flow and velocity distribution and pressure distribution of the gas.

FLUENT solver includes two types: serial and parallel solver. With the development of computational fluid dynamics, we often face the problems of large-scale computation and detailed research on the flow mechanism in engineering, which needs to divide the huge computation grid and the computation time is so huge. Since the calculated speed of using the serial solver is very slow and low efficiency, then we must use the parallel solver. Serial solver only needs one thread running and does computation on one CPU,while parallel solver needs make computational domain blocking partition, and deal with computing tasks using more than one process in divisional area [4-5].

In this paper, the solving speeds of serial and parallel solver with FLUENT software were compared, and the effect of different number of nodes on the speed of parallel computing was discussed. On this basis, the axial fan was simulated using three-dimensional numerical simulation and many important details, velocity distribution and pressure distribution inside the axial fan were obtained.

+ Corresponding author.
E-mail address: zhoujj@zzu.edu.cn

2. Model establishment and grid meshing

2.1. Subjects

This study for the R40 series fan [6], impeller diameter is 1.125m, tip hub ratio is 0.356.Standard air conditions, the flow rate Q equal to 13m^3/s, total pressure p_T equal to 140Pa, single impeller fan motor and impeller is directly connected, and the motor speed equal to 1440rpm.Impeller selection CLARK-Y airfoil and the relative thickness is 12%.

The fan blade section was defined by parameters of several cross sections and a distorted shape of fan blades was obtained by surface connected. Therefore, we used the modeling method of the point to a line and then to plane, to the body the modeling process[7].Solidworks model of the leaf blade in Figure 1.Fan overall model shown in Figure 2.

Fig1. Solidworks model of the leaf blade

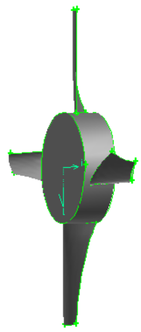


Fig.2. Fan overall model

2.2. Control Equations

the three-dimensional incompressible Reynolds-averaged Navier-Stokes Equations for additional turbulence equations is using time-dependent method As of heat exchange in the flow process is small can be neglected, therefore only apply continuity equation, momentum equations and k-ε equations, which was expressed as[7]:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{1}$$

Momentum conservation equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

k equation:

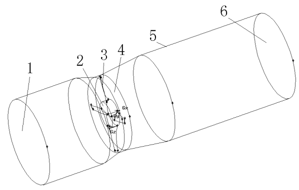
$$\rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G - \rho\varepsilon \tag{3}$$

ε equation:

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + \frac{c_1 \varepsilon}{k} G - c_2 \rho \frac{\varepsilon^2}{k} \tag{4}$$

2.3. Computational Domain and Mesh

In order to simulate the flow field inside the fan, we choose a relatively large computational domain, combined with the actual situation of fan flow; the fans were set up in front of and behind the converter assembly and diffuser. Schematic diagram of computational domain is shown in Figure 3.



1.inlet 2.Wheel 3.leaf blade 4.exteral cover 5.wall 6.outlet

Fig.3. Schematic diagram of computational domain

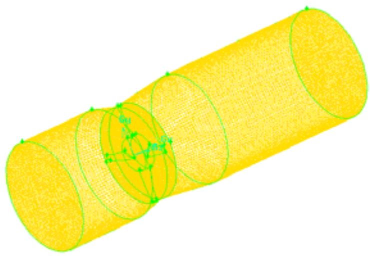


Fig.4. diagram of mesh in the domain

Determine the calculated domain was conducted and determine the meshing of domain the grid is shown in Figure 4.Boundary conditions is specified as follow: a given mass flow rate boundary conditions is used at inlet is; a free outlet boundary conditions is used at outlet; no slip solid wall boundary condition is used at wind tunnel wall, leaves blade and select Hub; fan rotating reference system coordinates multi-region model is used, a rotation speed keep constant; turbulence model is the Realizable modified k-ε model for swirl flow. Unrelaxation is used in Calculation process, the algorithm used SIMPLEC algorithm, Momentum Turbulent Kinetic Energy and Turbulent Dissipation Rate are using Second Order Upwind format.

3. Analysis and discussion

3.1. Comparison of Serial Calculation and Parallel Computation

Start FLUENT parallel Solver, the number of computing nodes is set at 2,the grid partitioning method with automatic meshing, meshing grid in the computation domain is shown in Figure 5,in order to facilitate the observation, the figure only shows the contour lines, figure 5 can show the oval region contour line in the middle part of the computation domain, and meshed grid in the flow field is divided into two parts, and two nodes setting of will be to solve these two parts and data exchange, respectively[8-10] .

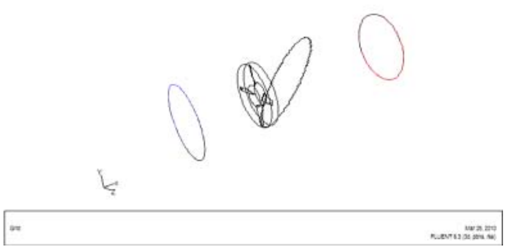


Fig.5. Grid map after segmentation

The computational results are compared using the serial and parallel solver at the same computation mesh in the section, respectively. the results shown in Table 1.the computational results of serial and parallel solver show that the total pressure of outlet calculated with the two solver is the same but the calculation speed with parallel solver is 39 times per minute, the calculation speed is significantly faster than that of the serial solver, that is to say, 23 times per minute. the iterative steps Calculated from starting calculation to converge by the two solver is almost same, the serial solver iterations is 1351 while 1348 for the parallel solver, due to faster computing, convergence time with parallel computation solver decreased by almost 40%.than that of the serial solver.

TABLE I. COMPARISON OF RESULTS CALCULATED WITH SERIAL AND PARALLEL SOLVER SOLVERTABLE CONTROL EQUATIONS

	Exit total pressure	Computing speed	Iterations	Convergence time
Serial Solver	134.84Pa	23times/minute	1351	59 minutes 35 seconds
Parallel Solver	134.93Pa	39 times/minute	1348	37 minutes 18 seconds

3.2. Comparison of Different Nodes with the Parallel Solver

By comparison of the previous section, you can see the apparent speed advantage of parallel solver in the calculation, two compute nodes is used for the above parallel computation in the previous section, and then there will be the two thread of FLUENT calculation at the same time. Each additional node, there will be increase a thread of FLUENT. The effect on calculation at different nodes is analyzed and discussed in this section. the number of nodes is Set 2,4,6,8,10 and 12,respectively,the results is shown in Table 2.The number of nodes stand for horizontal axis, and vertical coordinate axis stand for the convergence time, the plot is shown in Figure 6.

TABLE II. COMPARION OF RESULTS CALCULATION AT DIFFERENT NODES

Number of nodes	Exit total pressure	Computing speed	Iterations	Convergence time
2	134.93	39times/minute	1348	37 minutes 18seconds
4	134.62	40times/minute	1378	36minutes 32seconds
6	134.85	41times/minute	1403	35 minutes 43seconds
8	135.03	41times/minute	1442	36 minutes 07seconds
10	134.89	40times/minute	1470	36 minutes 54seconds
12	134.78	39times/minute	1482	37 minutes 53seconds

the results in table 2 show that the total pressure of outlet pressure is almost the same at different nodes number, when a node number fewer, with the increase number of nodes increases, the calculation speed slightly accelerated, and the convergence of iterations required increase at the same time. Generally, the more the number of compute nodes, the shorter the time required by parallel computing, the convergence time, however, is not very significant. This is because for the dual-core CPU computer, the set of two nodes for parallel computing, the computer CPU usage has been almost up to 100%,even if increase the number of nodes, but the increase of speed of calculation not obvious. When the number of nodes is higher, with the number of nodes increases, the calculation speed but decreased, because the number of nodes increases, data Communication between nodes also increased will also affect the speed of calculation. Once again if the calculation speed is to significantly improve; it is necessary to increase the number of CPU. Can be seen from Figure 6, for this example, when the number of nodes is six, calculating speed is the maxium, the convergence time calculated is the shortest, it will be used in six nodes parallel computing to improve computing efficiency in the following calculation.

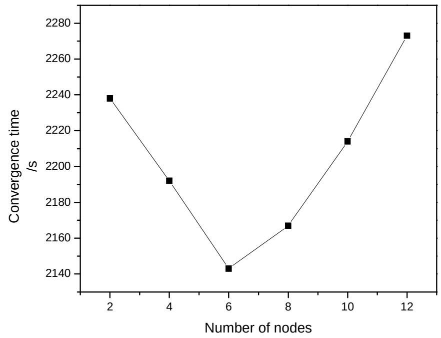


Fig.6. The curve of the convergence time and number of nodes

3.3. Fan Flow Field Analysis by Parallel Computing

The three-dimensional time averaged N- equations is solved to calculate the flow field for the axial fan at the design condition by the six nodes parallel solver.

(1) fan flow field analysis

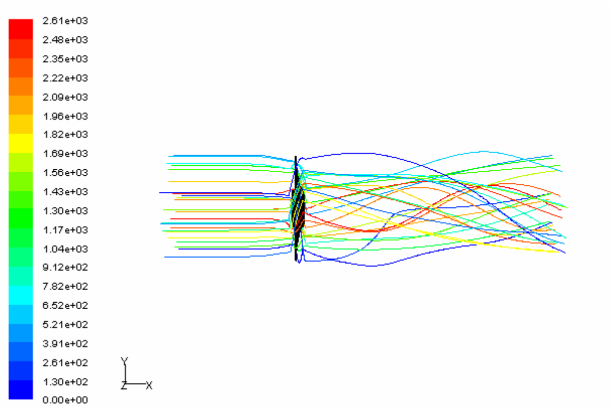


Fig.7.Trace chart for fan flow field

Figure 7 is the trace map of the flow patterns at the design conditions, from Figure 7 we can see: fluid flow traces at the fan outlet can be thought of as the spiral line with the wind turbine blades rotating, and the trace tangential direction from the outflow from the leaf blade is perpendicular to the trace tangential direction corresponding to the tangential point, the computation result are consistent with the analysis result adapted with the assumption of the velocity-triangle.

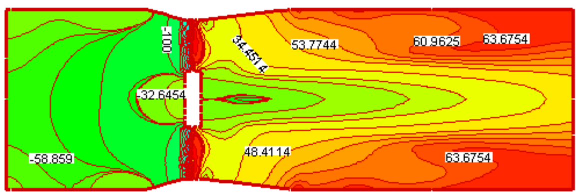


Fig.8. the pressure distribution for the fan flow field

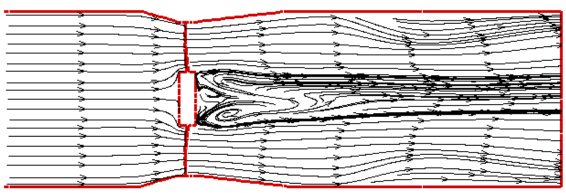


Fig.9. tail vortex of the fan

Figure 8 shows the pressure distribution of the fan center Z equal to 0, from the diagram seen, air flows through the impeller, a low pressure zone is formed of at the rear of the fan wheel, making the airflow from outlet of the fan flow to the zone of low pressure and generated vortex(shown in Figure 9).The vortex increases the air resistance of the flow at outlet, reduces the flow dynamic pressure of exit. If there is surplus space for the installation, Streamline cover to install, at the rear wheel can reduce drag effectively and improve efficiency of the fan.

(2) pressure distribution at blade surface

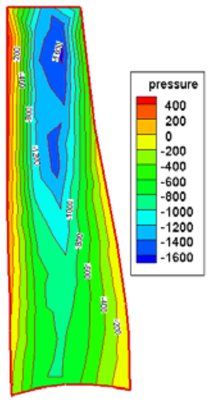


Fig.10. Static pressure distribution of blade suction surface

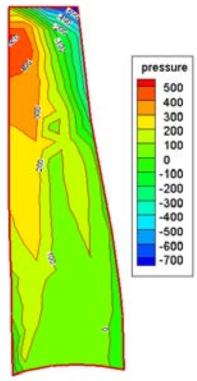


Fig.11.Static pressure distribution of blade surface

Figure 10 shows the static pressure distribution of blade suction surface, from the diagram seen, a radial pressure gradient is formed near the tip region, radial negative pressure gradient is formed near the blade root region, pressure gradient with the "C" shape is formed at the entire leaf surface, which is conducive to the chassis and wheels for the low energy fluid near the walls, breathe it leaves the central area of high energy to the mainstream, weaken the low energy fluid in the case and hub aggregation, thus to decrease flow losses, that reach the purpose of performance improving. Figure 11 shows the static pressure distribution of blade surface, we can see an obvious of low pressure area on the top of the wake regions in the leaves, which is due to the impact of tip clearance, and there are leaves at the top of the leakage flow. The presence of leakage vortex, increase the flow resistance losses, reducing the fan outlet pressure, which is prone to stall axial fan.

4. Conclusion

Compare with the serial and parallel solver of FLUENT software is conducted in the paper, three dimensional steady turbulent flow is used for numerical calculation of the flow, obtained with the practical situation of many flow details, draw the following conclusions:

(1) Parallel solver calculation speed is faster than that of the serial solver, the convergence time of the parallel solver decreased by almost 40%than that of the serial solver at the same grid.

(2)With the increase of the nodes number for parallel computation, the convergence time required decreased and then increased. For this example, when the number of nodes is equal to six, the calculation of the convergence time is shortest.

(3) When air flows through the impeller the vortex is generated at the rear of the wheel, which is important reason for reduced fan efficiency

(4)"C" type of pressure gradient distribution is formed at the surface of fan bending and torsion blades to improve the air flow distribution in the passage; there is leakage flow at the top of the trail, all that which increase resistance loss.

5. Acknowledgment

This work is supported by Henan Major public Scientific Research Projects (081100910100).

6. References

[1] Li Sheng.preliminary design of axial flow fan based on the FLUENT[D].Northwest Polytechnical University

[2] Zhang Wei,Wang Yuan.Experimental Study Progress of Internal Flow and Rotor/stator Interaction in Turbomachinery[J].FLUID MACHINERY,2001.

[3] L.Bermudez,et al.Numerical Simulation of Unsteady Aerodynamics Effects in Horizontal-Axis Wind Turbines[J].Solar Energy,2000,68(1):9-21.

[4] Han Zhan-zhong,Wang Jing,Lan Xiao-ping.FLUENT simulation of fluid engineering examples and applications [M].Beijing:Beijing Institute of Technology Press,2004.

[5] Wang Rui-jin,Zhang Kai,Wang Gang.FLUENT technology base and its application to[M].Beijing:Tsinghua University Press,2007.

[6] National Standardization Technical Committee of fans.GB3235-82 basic fan types,dimensions and characteristics of[S].Beijing:China Standard Press,1992.

[7] Huang Wen-jun,Li lu-ping.Research on three dimensional solid modeling for steam turbine twist blade based on the SlidWorks[J].Machinery Desien&Manufacture,2007,(5):67-69.

[8] Fu De-xun,Ma Yan-wen.Computational Fluid Dynamics[M]. eijing:Higher Education Press,2002.

[9] Wang Cheng-yao,Wang Zheng-hua,the ield of Energy.Computational fluid ynamics and parallel algorithms[M].unan:National Defense University ress,2000.

[10] Zhang Bao-lin,Gu Tong-xiang,Mo Ze-yang.Parallel computing theory and methods[M].Beijing:National Defence Industry Press,1999.