FLOW SIMULATIONS IN CROSS-FLOW FANS USING THE FINITE ELEMENT AND FINITE VOLUME METHODS

ANDRZEJ SOWA

Section of Fluid Mechanics Institute of Industrial Apparatus & Power Engineering, Cracow University of Technology, Al. Jana Pawła II 37, 31-864 Cracow, Poland
aisug@inetia.pl

(Received 8 July 2003; revised manuscript received 7 August 2003)

Abstract: Both basic computational fluid dynamics methods: the finite element method (FEM) and the finite volume method (FVM) have been used to simulate flow fields in a cross-flow fan (CFF). A review of previous numerical simulations of flow in CFF's is presented. The theoretical foundations of the applied numerical algorithms and specifications of the computer programs are given. The procedure of computations is described in detail. Computational results are shown in the form of contour and vector velocity and contour pressure plots.

Keywords: turbomachinery, cross-flow fan, finite element method, finite volume method, usteady flow analysis

Nomenclature

a_0 – the speed of sound,
C_p – the pressure coefficient,
E – the bulk modulus,
M – the Mach number,
p – pressure,
u – velocity,
u_n – velocity component normal to boundaries,
λ – the volume viscosity coefficient,
μ – dynamic viscosity,
ν – kinematic viscosity,
ρ – density.

1. Introduction

Cross-flow fans (CFF) belong to a unique type of turbomachines whose mode of operation is completely different from that of the commonly used fans, either axial or centrifugal.

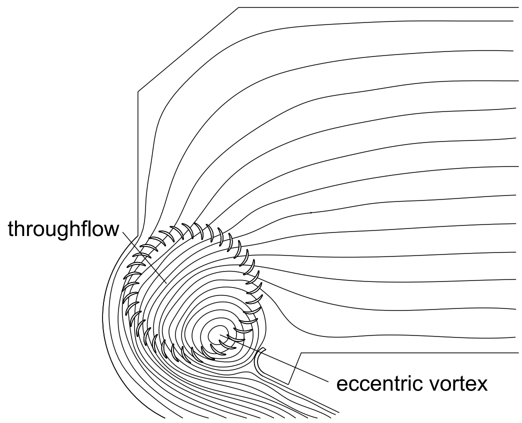

Figure 1. Main zones of the flow area inside a CFF, Moon *et al.* [1]

The area of internal flow inside a CFF has usually been divided into two regions (Figure 1):

A. the eccentric vortex region, characterized by closed streamlines, also called the recirculating region,
B. the throughflow region, being the main asymmetrical flow in the interior of the fan.

Even though plenty of experimental research and some advanced mathematical analyses of this fascinating but complicated flow phenomenon in CFF's have been carried out [2], there is no generally accepted method for their design, mainly because of the complexity of flow conditions: the unsteady and highly turbulent nature of the twice accelerated fluid, the cyclic and variable working conditions of the blades, as well as the continuous crossing of the main eccentric vortex by the cascade of blades.

The finite difference volume method and the finite difference element method are the core of computational fluid dynamics (CFD), a relatively young and spontaneously developing branch of science. Because of the necessity to implement complicated numerical algorithms and discretization methods to the unsteady, turbulent flow in an irregular casing, there have been only a few attempts of numerical simulation of this turbomachine. Some of them, made for selected geometries and operating conditions, have been successful [3], which encourages us to search for a more exact and complete numerical analysis of that interesting fan.

2. Review of previous numerical simulations

The first numerical simulations of fluid flow inside a cross-flow fan were carried out by Harloff and Wilson [4]. They applied the finite element method (FEM) for a two-dimensional, steady, incompressible flow of an inviscid fluid. The simulation was restricted to throughflow.

The boundary element method (BEM) was used by Kitagawa *et al.* [5] to simulate the 2D potential and the transient flow of air. As a result, a shedding vortex visualization over the entire flow area was obtained.

In 1996 Bert *et al.* [6] computed turbulent steady and unsteady flows by means of the finite element method which included the whole cross-section of the fan. They assumed the incompressible flow model and, as a consequence, arrived at a sensible approximation of the real streamlines.

Yeh's calculations [7] are one of the best flow simulation of cross-flow fans performed so far. He used the finite element method assuming a turbulent, unsteady and weakly compressible flow of a viscous fluid.

The boundary element method was used by Tsurusaki *et al.* [8]. In contrast to the assumptions made by Kitagawa *et al.* the random walk model was applied instead of the vortex decay model.

In 1998 Korean researchers Moon *et al.* [1], simulated the two-dimensional unsteady incompressible laminar flow of a viscous fluid achieving a high level of consistency between the calculated flow fields and the experimental ones.

Two-dimensional computations of the unsteady incompressible and viscous flow were carried out by Dornstetter and Gabi [9], using the finite volume method (FVM) code. Satisfactory agreement between numerical and experimental results was reached.

A fully implicit finite volume method solving the Reynolds-averaged Navier-Stokes viscous differential equations was employed by Bin *et al.* [10] to investigate the three-dimensional unsteady internal flow field of two cross-flow fans: a traditional straight CFF and a twist one.

In Poland, the first successful simulations of fluid flow inside a cross-flow fan were carried out by author of this paper using finite element and finite volume programs. The theoretical basis of the computations and the numerical results in the form of graphic plots are presented below.

3. Numerical algorithms and programs

3.1. YFLOW-Rota code

The Taiwanese YFLOW-Rota 2D solver is a CFD program which works by means of the finite element method. This algorithm allows one to calculate unsteady compressible/hydrodynamic (weakly compressible) turbulent flows of viscous fluids in turbomachines, so it has all the features needed to simulate a real flow field in a CFF. The following form of the Navier-Stokes equations, modified by He and Song [11], have been used in the algorithm.

3.1.1. Continuity equation

The equation of state for weakly compressible flow represented by:

$$p-p_0=a_0^2(\rho-\rho_0), \tag{1}$$

substituted into the compressible continuity equation,

$$\frac{\partial\rho}{\partial t}+u_i\frac{\partial\rho}{\partial x_i}+\rho\frac{\partial u_i}{\partial x_i}=0 \qquad (i=1,2), \tag{2}$$

produces the following relationship:

$$\frac{\partial p}{\partial t}+u_i\frac{\partial p}{\partial x_i}+\rho_0 a_0^2\frac{\partial u_i}{\partial x_i}=0. \tag{3}$$

Using dimensional analysis, Song and Yuan [12] proved that the second term in Equation (3) is of the order of M^2 and may thus be neglected for a weakly compressible flow. Thus, the transformed continuity equation takes this final form:

$$\frac{\partial p}{\partial t}+E\frac{\partial u_i}{\partial x_i}=0. \tag{4}$$

3.1.2. Momentum equation

The equation of motion for a weakly compressible flow is the same as that for an incompressible fluid, that is:

$$\rho_0\left(\frac{\partial u_i}{\partial t}+\frac{\partial}{\partial x_j}(u_j u_i)\right)=-\frac{\partial p}{\partial x_i}+\mu\frac{\partial^2 u_i}{\partial x_j \partial x_j} \qquad (i,j=1,2), \tag{5}$$

where μ is the sum of dynamic viscosity, μ_d, and Smagorinsky's eddy viscosity, μ_t, used for modeling small-scale vortices in the large eddy simulation (LES) model.

Smagorinsky's turbulent viscosity may be written as:

$$\mu_t=\frac{1}{2}\rho(C_s\Delta)^2\left|\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right|. \tag{6}$$

The length scale, Δ, was assumed as the square root of the filter area and the calibration coefficient, $C_s=0.17$, was chosen as the middle value of the recommended range: 0.1–0.24 [13].

The following velocity-dependent pressure boundary conditions were assigned at the inlet/outlet of the blower:

$$p=\frac{1}{2}\rho C_r|u_n|u_n. \tag{7}$$

During the computations the coefficient of flow resistance was set as $C_r=1$ at boundaries. Owing to small time increments (order of magnitude about 10^{-6}s), there was no risk of exceeding the Courant-Friedrich-Lewy criterion of stability, so an explicit time discretization scheme was employed, which does not require solving an additional system of equations.

The sliding mesh technique has been applied to ensure satisfactory compatibility of interface between the rotating domain of the impeller and the fixed outside area.

3.2. Flo++ program

The algorithm of the Flo++ solver uses the finite volume method. This 3D CFD program has the capability of calculating compressible and incompressible, unsteady as well as turbulent flows of viscous fluids, so it appears to be a tool applicable to simulation of flows in CFF's. The standard Reynolds k-ε high turbulence model was employed. Unsteady equations were solved by the fully implicit scheme and the transient PISO algorithm. The transient PISO is a non-iterative procedure, so there was no need to use underrelaxation to improve stability of the calculation process. The linear equation system was solved by the biconjugate gradient method, which is fast and effective.

3.3. Details of the simulations

3.3.1. YFLOW-Rota program

ANSYS/Multiphysics Research FS 5.7 was used as a pre/postprocessor with a YFLOW-Rota solver. The geometry of the fan, the computational grid and the boundary conditions were defined in ANSYS. The rotational speed of the impeller, flow type, fluid parameters, the coordinates of the calculated points of pressure and velocity, as well as the total time of calculations were written in separate input files. The Plane 42, that is a flat element, were selected, since comparative calculations had proved their advantage over the triangular one [14]. A segment of the non-uniform grid with quadrilateral elements is shown in Figure 2.

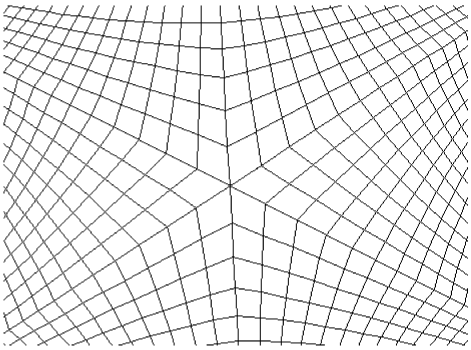

Figure 2. Quadrilateral elements of the grid

The input files were generated by the ANSYS preprocessor and read by the YFLOW solver working in the DOS prompt mode. The simulation required about 70 hours of calculations on a PC with a Pentium III 1GHz CPU and 256MB of RAM. As a result of the calculations, YFLOW-Rota output files containing the component values of nodal velocity, static pressure at particular elements, and the flow rate values in the inlet and outlet regions of the CFF were obtained. A visualization of static pressure and velocity was arrived at after conversion in the ANSYS postprocessor. It was possible to display contour and vector velocity plots, as well as contour pressure plots in the postprocessor. The Ansys Animate Utility allowed us to create their animations, too.

3.3.2. Flo++ program

Flo++ is a complete CFD program with a pre/postprocessor and a solver. The input data for its preprocessor were written in an input file with Flo++ Command Language. The geometry was drawn in Microstation/J and inserted into the Flo++ preprocessor in the form of an IGES file.

The computational area was divided into three zones: the exterior one with an inlet and an outlet, the cascade of blades region and the interior zone. The cross section of the fan was split into blocks, and block-structured grids were build for complex geometries with non-matching grids at interfaces (the arbitrary mesh coupling technique).

The finite volume element computations are three-dimensional by definition but because, in principle, the nature of the flow in a CFF is invariable along its axis, a single thin layer of tetrahedral cells perpendicular to the axis was used with the

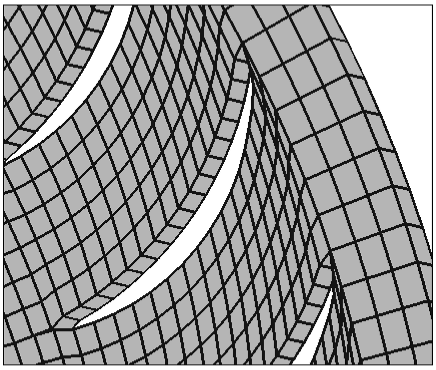

Figure 3. A fragment of the cascade with tetrahedral elements

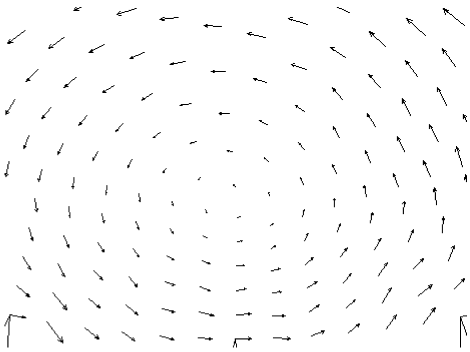

Figure 4. The eccentric vortex region near the internal edge of the blading

two-sided symmetry boundary condition (Figure 3) and the computed flow can be regarded as a plane one.

Constant pressure at the inlet and outlet boundaries as well as the logarithmic wall function, for the high-turbulent flow model, were assigned as boundary conditions.

The sliding mesh technique was applied for the impeller-casing interaction. Unsteady flow was realized by setting the revolutions at a desired frequency and automatically adjusting the time step in order to meet the criteria of the pre-set Courant number.

The simulation of six revolutions of the impeller required a total of more than 100 hours of computing time. It was necessary to write the input file containing commands for the postprocessor in Flo++-interpreted language. As a result, plots of velocity as well as pressure were generated.

4. Numerical results

4.1. Velocity field

The proper flow structure, with all characteristics of CFF flow zones, was obtained as a result of computations by the YFLOW-Rota code. The eccentric vortex region adjacent to the stabilizer is shown in Figure 4.

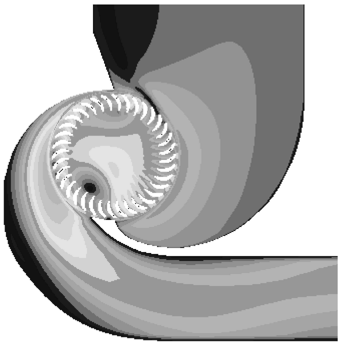

Figure 5. The contour velocity field; brighter tints represent higher velocities

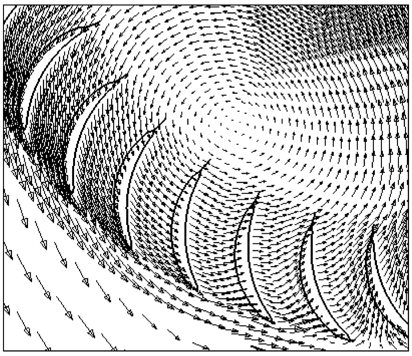

Figure 6. A segment of the velocity vector field with the eccentric vortex

A correct flow field in the cross-flow fan was also generated by the Flo++ program, and the distribution of velocities is given in the form of contour (Figure 5) and vector (Figure 6) plots.

4.2. Contour pressure plots

For the reader to better understand the next plot it is necessary for us to give a description of the formation process of the main flow structures in the CFF first (on the basis of the numerical results). Small vortices are shed from the blade system right after starting the impeller. Vortices which are shed inside the rotor merge and after some time there are only a few left, which can be observed as closed isobars in Figure 7 (which is a contour pressure plot drawn for the moment corresponding to two revolutions of the rotor). The throughflow arises at the same time. Then separate vortices integrate and a big vortex arises and moves towards the blading, forming the eccentric vortex. Small vortices are continually shed from the blades, even when the steady-state is achieved. Some vortices flow outside the impeller, which is the cause of the great rotation of fluid particles at the outlet. It may be supposed that there is a 3D turbulence in the discharge channel but no observation of this phenomenon was possible by means of 2D computations. A more detailed description of the process may be found in paper [15].

The eccentric vortex region obtained from the calculations using the Flo++ program is shown in Figure 8. It can be seen as a region characterized by the lowest values of static pressure (the darker region).

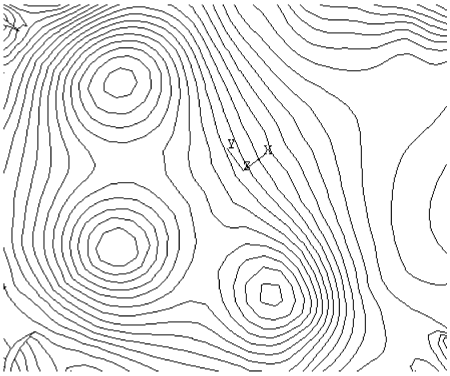

Figure 7. Eccentric vortex formation from smaller vortices, YFLOW-Rota code

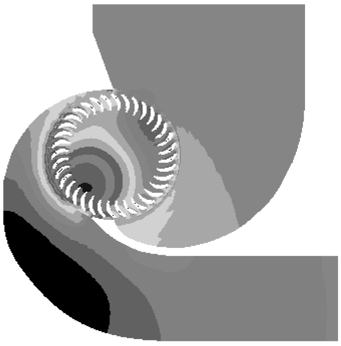

Figure 8. The static pressure contour plot including the eccentric vortex region

5. Summary

Numerical simulations of flow in a very special turbomachine such as a cross-flow fan could be performed owing to significant progress in applied mathematics, computer science and numerical fluid dynamics. Although simulations cannot be a substitute for real physical experiments, they can be regarded as a very important complementary method, particularly where experiments are difficult to perform or prohibitively expensive.

The information about the internal flow field in a cross-flow fan obtained by these computational fluid dynamics methods is most useful and seems to be an essential step towards understanding the complex fluid flow phenomena in CFF's. As such, it may enable further improvement in their design and the elaboration of a general and coherent analytical model of this unique machine.

References

[1] Moon Y J, Cho Y and Nam H 1998 *Proc. 4th KSME-JSME Fluid Engng. Conf.*, paper-006

[2] Stacharska-Targosz J and Sowa A 2001 *Technical J., Series of Mechanics*, Cracow University of Technology Publishing, Cracow, pp. 1–20 (in Polish)

[3] Sowa A 2001 *Technical J., Series of Mechanics*, Cracow University of Technology Publishing, Cracow, pp. 117–125 (in Polish)

[4] Harloff G J and Wilson D R 1981 *Aircraft* **18** (4) 310

[5] Kitagawa K, Tatsuke H, Tsujimoto Y and Yoshida Y 1992 *Int. Conf. Computer Modeling of Seas and Coastal Regions and Boundary Elements in Fluid Dynamics* (Brebbia C A and Partridge P W, Eds), Computational Mechanics Publications, Southampton, pp. 3–20
[6] Bert P F, Pessiani M, Combes J F and Kueny J L 1996 *Int. Gas Turbine and Aeroengine Congress & Exhibition*, Birmingham, pp. 1–10
[7] Yeh J 1996 *Aerothermodynamics of Internal Flows III* (Shen Yu, Naixing Chen and Yinming Bai, Eds), World Publishing Corp., pp. 773–779
[8] Tsurusaki H, Tsujimoto Y, Yoshida Y and Kitagawa K 1997 *J. Fluid Engng.* **119** 633
[9] Dornstetter S and Gabi M 2001 *Proc. 5th Int. Symposium on Experimental and Computational Aerothermodynamics of Internal Flows*, Gdansk, Poland, pp. 117–125
[10] Bin Y, Elhadi E and Keqi W 2002 *Proc. 4th Int. Conf. Pumps and Fans*, Tsinghua University, Beijing, pp. 344–351
[11] He J and Song C C S 1993 *Engineering Applications of Large Eddy Simulations ASME* **162** 147
[12] Song C C S and Yuan M 1998 *J. Fluids Engng.* **110** 441
[13] Tannehill J C, Anderson D A and Pletcher R H 1984 *Computational Fluid Mechanics and Heat Transfer*, Taylor & Francis
[14] Matuszyk P J 2000 *Solution of 2D Steady Heat Transfer Equation by Finite Element Method*, MSc Thesis, AGH, Cracow, Poland (in Polish)
[15] Stacharska-Targosz J and Sowa A 2002 *XV National Conf. Fluid Mechanics*, Augustow, Poland, CD-ROM Collection of Papers (in Polish)