An Adaptive Turbulence Model for Swirling Flow

Buntić Ogor, I.*, Gyllenram, W., Ohlberg E.*, Nilsson H.**, Ruprecht A.***

* Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Germany

** Div. of Fluid Dynamics, Dept. of Applied Mech., Chalmers University, Göteborg, Sweden

ABSTRACT

Swirling flows are very dominant in applied technical problems, especially hydraulic machinery, and their prediction requires rather sophisticated modelling. At present an applicative method for simulation is Very Large Eddy Simulation (VLES). In VLES large turbulence structures are resolved by an unsteady simulation and the minor structures are modelled with an adequate turbulence model. Therefore turbulence model must distinguish between resolved and unresolved scales. The VLES method also introduces a filtering technique which helps the turbulence model to adapt in accordance with the scales to be modelled. As a basis the modified k-ε model of Chen and Kim used with additional streamline curvature correction of Reif. The model is implemented in both FENFLOSS and CALC-PMB CFD codes which are used for simulation of swirling pipe flow and swirling flow through a straight conical diffuser, respectively.

INTRODUCTION

Flows which appear in different technical problems are characterised as very intricate turbulent flows followed with different flow phenomena, e.g. unsteadiness, swirling flow, separation of the flow etc. Thus their simulations are complicated and time consuming requiring high computational power. Additionally, adequate turbulence modelling is needed which is able to predict these flows satisfactorily.

Turbulence modelling is still one of the fundamental problems of Computational Fluid Dynamics (CFD). Application of classical Reynolds-averaged Navier-Stokes (RANS) simulation with usual turbulence models, e.g. k-ε or k-ω model, often gives inadequate results. The highest accuracy for resolving complete turbulence is offered by a Direct Numerical Simulation (DNS). Unfortunately its industrial application is not possible in the foreseeable future due to the fact that it requires extremely find grid resolution for performing 3D simulation of the flow with high Reynolds number.

Lately Large Eddy Simulation (LES) starts to be a mature technique despite its necessity for high computational resources. With LES all anisotropic turbulent structures are resolved in the computation and only the smallest isotropic scales are modelled with the models which are simpler compared to those used for RANS.

At present an applicative method for simulation is Very Large Eddy Simulation (VLES). It is a kind of hybrid method which starts to expand as a promising compromise for simulation of industrial flow problems with reasonable computational time and costs. In VLES large turbulence structures are resolved by an unsteady simulation and the minor structures are modelled with an adequate turbulence model.

SIMULATION METHOD

Governing equations and turbulence modelling

The governing equations describing incompressible, viscous and time dependant flow are the Navier-Stockes equations. In the RANS approach, these equations are time or ensemble averaged leading to

$$\frac{\partial \overline{U}_i}{\partial t} + \overline{U}_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_j} + \nu \nabla^2 \overline{U}_i - \frac{\partial \tau_{ij}}{\partial x_j} \qquad (1)$$

$$\frac{\partial \overline{U}_i}{\partial x_i} = 0 \,. \qquad (2)$$

Here the Reynolds stress tensor is unknown and the task of turbulence modelling is the formulation and determination of suitable relations for Reynolds stresses.

RANS equations are established as a standard tool for industrial simulations, although it means that the complete turbulence behaviour has to be enclosed within appropriate turbulence model which takes into account all turbulence scales (from the largest eddies to the Kolmogorov scale). Up till now the mostly used turbulence models are standard k-ε, k-ω or their variations. They are developed for modelling the whole range of turbulent scales and it is well known that they show excessive viscous behaviour very often damping the unsteady motion quite early.

Lately several hybrid methods are proposed in the literature and among them VLES. They are all based on the same idea to represent a link between RANS and LES. They try to keep computational efficiency of RANS and the potential of LES to resolve large turbulent structures, even on coarser grids and with high Reynolds number. Their main difference compared to the LES is that a smaller part of the turbulence spectrum is resolved and the influence of a larger part of the spectrum has to be expressed with the model (Fig. 1). Additional requirement is appropriate filtering technique which distinguishes between resolved and modelled part of the turbulence spectrum. It provides their adaptive characteristic enabling them to be applied for the whole range of turbulence modelling approaches from the RANS to the DNS (Fig. 2).

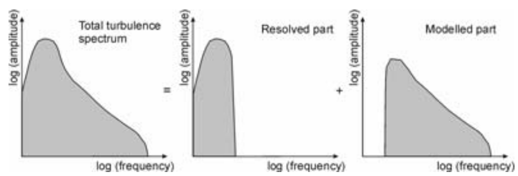

Figure 1: *Modelling approach used in VLES.*

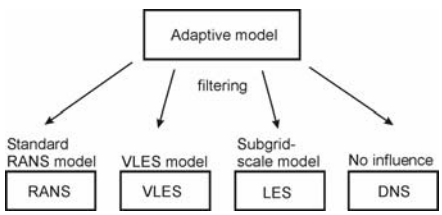

Figure 2: *Principle of filtering and adjustment for adaptive model.*

The basis of the adaptive model is the extended k-ε model of Chen and Kim [1]. It is chosen due to its simplicity and capacity to better handle unsteady flows compared to the standard k-ε model. Its transport equations for k and ε are given as

$$\frac{\partial k}{\partial t}+\overline{U}_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\varepsilon \quad (3)$$

$$\begin{aligned}\frac{\partial \varepsilon}{\partial t}+\overline{U}_j\frac{\partial \varepsilon}{\partial x_j}&=\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\\ &c_{1\varepsilon}\frac{\varepsilon}{k}P_k-c_{2\varepsilon}\frac{\varepsilon^2}{k}+\underbrace{c_{3\varepsilon}\left[\frac{P_k}{k}\right]\cdot P_k}_{\text{additional term}}\end{aligned} \quad (4)$$

with following coefficients:
$\sigma_k = 0.75$, $\sigma_\varepsilon = 1.15$, $c_{1\varepsilon} = 1.15$, $c_{2\varepsilon} = 1.15$ and $c_{3\varepsilon} = 0.25$.

Additionally, these extended k-ε equations need to be filtered. Applied filtering technique is similar to Willems [2]. The smallest resolved length scale Δ used in filter is according to [3] dependant on the local grid size or the computational time step and local velocity.

According to the Kolmogorov theory it can be assumed that the dissipation rate is equal for all scaled. This leads to

$$\varepsilon=\hat{\varepsilon} \quad (5)$$

It is not acceptable for turbulent kinetic energy. It is filtered according to

$$\hat{k}=k\left[1-f\left(\frac{\Delta}{L}\right)\right] \quad (6)$$

As a suitable filter

$$f=\begin{cases}0 & \text{for} \quad \Delta \ge L\\ 1-\left(\frac{\Delta}{L}\right)^{2/3} & \text{for} \quad L>\Delta\end{cases} \quad (7)$$

is applied where

$$\Delta=\alpha\cdot\max\begin{Bmatrix}|u|\cdot\Delta t\\ h_{\max}\end{Bmatrix} \quad \text{with} \quad h_{\max}=\begin{cases}\sqrt{\Delta V} & \text{for 2D}\\ \sqrt[3]{\Delta V} & \text{for 3D}\end{cases} \quad (8)$$

contains model constant α in a range from 1 to 5. Then the Kolmogorov scale L for the whole spectrum is given as

$$L=\frac{k^{3/2}}{\varepsilon}. \quad (9)$$

Modelled length scales and turbulent viscosity are

$$\hat{L}=\frac{\hat{k}^{3/2}}{\hat{\varepsilon}} \quad (10)$$

$$\hat{\nu}_t=c_\mu\cdot\frac{\hat{k}^2}{\hat{\varepsilon}} \quad (11)$$

with $c_\mu = 0.09$.

The filtering procedure leads to the final equations

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\hat{\nu}_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+\hat{P}_k-\varepsilon \quad (12)$$

$$\begin{aligned}\frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}&=\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\hat{\nu}_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\\ &c_{1\varepsilon}\frac{\varepsilon}{k}\hat{P}_k-c_{2\varepsilon}\frac{\varepsilon^2}{k}+c_{3\varepsilon}\left[\frac{\hat{P}_k}{k}\right]\cdot\hat{P}_k\end{aligned} \quad (13)$$

with the production term

$$\hat{P}_k=\hat{\nu}_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}. \quad (14)$$

For more details of the model and its characteristics the reader is referred to [4].

The streamline curvature correction of Reif ([5]) is also introduced. In this model the Bousinesq hypothesis is generally defined by making the constant c_μ depending on the strain rate and the rotation rate tensors.

Numerical methods

<u>*FENFLOSS*</u>

FENFLOSS (Finite Element based Numerical FLOw Simulation System) is Finite Element Method based CFD code which is developed at the Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart. It uses 8-node hexahedral elements for spatial domain discretisation and the time discretisation involves a three-level fully implicit finite difference approximation of 2nd order. For the velocity components and the turbulence quantities a trilinear approximation is applied and the pressure is assumed to be constant within element. For advection dominated flow a Petrov-Galerkin formulation of 2nd order with skewed upwind orientated weighting function is used.

For the solution of the momentum and continuity equations a segregated algorithm is used. The equations are linearised and the linear system is solved with a conjugated gradient method BICGSTAB2 with an incomplete LU decomposition (ILU) for preconditioning. The pressure is treated with the modified Uzawa pressure correction scheme [6], which is performed in an inner iteration loop without reassembling the system matrices until the continuity error is reduced to a given order.

After the turbulence quantities are calculated and a new turbulence viscosity is gained. The equations of turbulence model are also linearised and solved with BICGSTAB2 algorithm. The whole procedure is carried out in a global iteration until convergence is obtained. For unsteady simulation the global iteration has to be performed for each time step.

The code is parallelised ([7], [8]) and computational domain is decomposed using overlapping grids. The linear solver BICGSTAB2 has a parallel performance and the data exchange between the domains is organised on the level of the matrix-vector multiplication using MIP (Message Passing Interface) on computers with distributed memory and OpenMP on the shared memory computers.

CALC-PMB

CALC-PMB CFD software is developed at the Division of Fluid Dynamics, Department of Applied Mechanics at Chalmers University of Technology, Göteborg. This in-house code is based on the finite volume method and the pressure-velocity coupling is solved using the SIMPLEC algorithm developed by van Doormaal [9]. Conformal block-structured, boundary-fitted coordinates are used and the code is parallelized for three-dimensional flows by domain decomposition. MPI is used for the exchange of information between the different processes/blocks, and two ghost cells are employed at the block interfaces to enable different first and second order discretisation schemes. The principal unknowns are the Cartesian velocity vector components (U,V and W) and the pressure (P). To avoid spatial oscillations of the pressure field over the collocated (non-staggered) grid arrangement, Rhie & Chow interpolation is applied for convections through the cell faces. For the discretised (linearised) system of equations, TDMA and conjugated gradient methods is implemented as the standard algorithms. For any further details the reader is referred to [10].

APPLICATIONS

For testing the performance of before mentioned models two test cases are chosen. The first test case is swirling flow in the straight pipe and its experimental data are available by Steenbergen ([11]). The second test case is a swirling flow through the straight conical diffuser. Experimental data by Clausen ([12]) are used for model validation. Both test cases are included in ERCOFTAC database.

Swirling pipe flow

According to the experiment settings, the computational domain is reconstructed as 3D pipe with constant diameter of 0.32 m. Available measurements are used for setting correct boundary conditions. The first section of measurements (three velocity components and Reynolds stresses) is used at the inlet. The considered Reynolds number is 300 000 and initial swirl intensity $S_0 = 0.18$.

Computations are carried out with standard k-ε model, k-ε of Chen and Kim, as well as with VLES. As expected standard k-ε model shows very poor results in case of intensive swirling flow. Model of Chen and Kim shows its known less damping characteristic, while VLES manages with unsteady calculations to catch and resolve clear unsteady vortex motion. Fig. 3 shows the comparison of the pressure fields calculated with standard k-ε model, k-ε model of Chen and Kim and VLES.

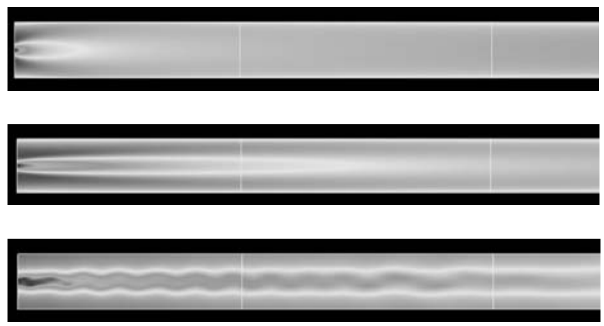

Figure 3: *Pressure filed of unsteady vortex in straight pipe calculated with standard k-ε model (up), extended k-ε model of Chen and Kim (middle) and VLES (down).*

Swirling flow through a straight conical diffuser

CFD code CALC-PMB was used for simulation of swirling flow through a straight conical diffuser. The diffuser has a half opening angle of 10° and the Reynolds number of the flow is 202 000. The swirl number is 0.59. In the experiment, the exit of the diffuser was open to the atmosphere. In the calculations, a large expansion is located at the diffuser exit in order to simulate similar outlet boundary conditions.

The filtering technique which allows the existence of large scale turbulence in the solution of the momentum equations while modelling small scale turbulence is applied to the standard high Reynolds number (HRN) k-ε model. Also the k-ε model of Reif et al. and k-ε model of Chen and Kim are investigated.

Looking at the time-averaged results, there is hardly any difference between the four versions of the k- ε model. However, they all differ with respect to resolved unsteadiness, visible in the instantaneous solutions. In the case where the filtered k-ε model has been used, the solution near the diffuser outlet is characterized by random turbulence. The solutions obtained from using the model of Reif et al. suggest helicoidal vortex filaments at the same location. The model of Reif et al. seems to allow a higher degree of secondary flow in the diffuser. Streamwise vortex filaments can be found along the walls of the expansion. The vortices are visualized by iso-surfaces of the second invariant of the strain rate tensor (Fig. 4).

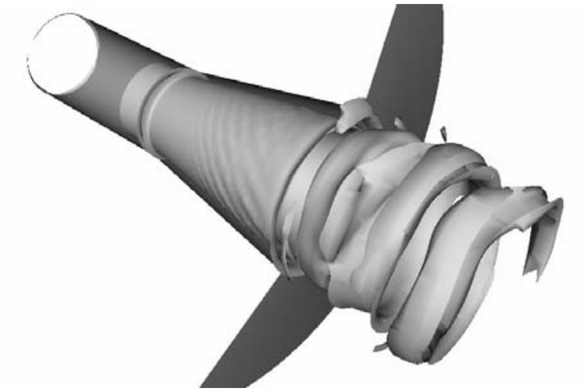

Figure 4: *Iso-surfaces of the second invariant of the strain rate tensor. Helicoidal vortex filaments are found near the diffuser exit. The turbulence model of Reif et al. has been used.*

CONCLUSIONS

For strong swirling flows the use of the standard k-ε model leads to rather poor results. By the application of a VLES approach based on the extended k-ε model of Kim and Chen with the additional streamline curvature correction of Reif the quality of the predictions can be significantly improved.

ACKNOWLEDGEMENTS

The research presented in this paper by Mr. Gyllenram has been a part of the “Water turbine collaborative R&D program” which is financed by Swedish Energy Agency, Hydro Power companies (through Elforsk AB), GE Energy (Sweden) AB and Waplans Mekaniska Verkstad AB. The computational work was carried out under the HPC-EUROPA project (RII3-CT-2003-506079), with the support of the European Community - Research Infrastructure Action under the FP6 “Structuring the European Research Area” Programme.

BIBLIOGRAPHY

[1] Chen Y.S. and Kim S.W. “*Computation of turbulent flows using an extended k-ε turbulence closure model”* NASA CR-179204, 1987.

[2] Willems W. „*Numerische Simulation turbulenter Scherströmungen mit einem Zwei-Skalen Turbulenzmodell“* Ph.D. thesis, Shaker Verlag, Aachen, 1997.

[3] Magnato F. and Gabi M. “*A new adaptive turbulence model for unsteady flow fields in rotating machinery”* Proceedings of the 8th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC 8), 2000.

[4] Ruprecht A. “*Numerische Strömungssimulation am Beispiel hydraulischer Strömungsmaschinen”* Habilitation thesis, University of Stuttgart, 2005.

[5] Pettersson Reif B.A., Abrahamsen P. E. H., Andersson H. I. and Solbakken S. *“ Modelling Rotational Effects in Industrial Closure Models”* 2nd International Symposium on Turbulence and Shear FLow Phenomena, Stockholm, Sweden, 2001.

[6] Ruprecht A. „*Finite Elemente zur Berechnung dreidimensionaler turbulenter Strömungen in komplexen Geometrien“* Ph.D. thesis, University of Stuttgart, 1989.

[7] Maihöfer M. „*Effiziente Verfahren zur Berechnung dreidimensionaler Strömungen mit nichtpassenden Gittern*“ Ph.D. thesis, University of Stuttgart, 2002.

[8] Maihöfer M. and Ruprecht A. “*A Local Grid Refinement Algorithm on Modern High-Performance Computers”* Proceedings of Parallel CFD 2003, Elsavier, Amsterdam, 2003.

[9] Van Doormaal J.P. and Raithby G.D. *“Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows”* Num. Heat Transfer, Vol. 7, pp. 147-163, 1984.

[10] Nilsson H. *“Numerical Investigations of Turbulent Flow in Water Turbines”* Ph.D. thesis, Dept. of Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 2002.

[11] Steenbergen W. *“Turbulent Pipe Flow with Swirl”* Ph.D. thesis, Eindhoven University of Technology, Eindhoven, 1995.

[12] Clausen P.D., Koh S. G. and Wood D. H. *“Measurements of a Swirling turbulent Boundary Layer Developing in a Conical Diffuser”* Experimental Thermal and Fluid Science, Vol. 6, pp. 39-48, 1993.