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 common in technical applications, especially in hydraulic machinery, and they require rather sophisticated modeling. At present, an applicative method for simulating unsteady flows is Very Large Eddy Simulation (VLES). In VLES, large turbulence structures are resolved in time and space, while the small scales are modeled with an adequate turbulence model. Turbulence model must therefore be able to distinguish between the resolved and the unresolved scales. In order to accomplish this, the method used in this work employs an adaptive and dynamic filtering technique, which restricts the turbulence model to only predict the effect of non-resolvable turbulent scales. The results obtained using the standard k-ε model is compared to those using the extended model of Chen and Kim with and without the filtering approach. The modified k-ε model of Reif et al. is also investigated in combination with the filtering approach. The models are implemented in the FENFLOSS and the CALC-PMB CFD codes. The chosen test cases are swirling flow in a straight pipe and swirling flow through a straight conical diffuser.

INTRODUCTION

Swirling, unsteady and separated turbulent flows are found in many technical applications and they are often intricate and complex to investigate. Numerical simulations of such flows are very time consuming and require high computational power. Additionally, adequate turbulence modeling is crucial in obtaining an accurate and satisfactory solution.

Turbulence modeling is still one of the fundamental problems of Computational Fluid Dynamics (CFD). Application of the classical Reynolds-averaged Navier-Stokes (RANS) simulation with the standard 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, DNS is in the foreseeable future not an option for industrial application, due to the fact that it requires extremely fine grid resolution and is therefore very computationally costly.

Lately, Large Eddy Simulation (LES) has begun to mature as an applicable technique, despite its need for very large computational resources. With LES, all anisotropic turbulent structures are resolved in time and space, leaving only the smallest scales to be modeled usually using very simple turbulence models. However, LES is still time consuming and immoderate for most engineering flow applications.

At present, one of the most applicative methods for simulation of turbulent flows is Very Large Eddy Simulation (VLES). It can be classified as a hybrid method and is a promising compromise between RANS and LES for the simulation of industrial flow problems. In VLES large turbulence structures are resolved in an unsteady simulation and the small structures are modeled with an adequate turbulence model. Consequently, VLES requires more sophisticated turbulence models than LES, but it still can be made at reasonable computational time and cost.

SIMULATION METHOD

Governing equations and turbulence modeling

The governing equations for incompressible, viscous and time-dependent flow are the incompressible Navier-Stokes equations. In the RANS approach, these equations are time or ensemble averaged. The averaging procedure leads to the following form:

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

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

Here, the Reynolds stress tensor, τ_{ij}, is unknown and has to be modeled. Until now, the mostly used turbulence models have been the standard k-ε and k-ω models or their variations. They are developed for modeling the whole range of turbulent scales, from the largest eddies to the Kolmogorov scales, and it is well known that they show excessive viscous behavior. In unsteady simulations, they usually tend to damp the unsteady motion quite early, which often results in steady solutions.

Recently, several hybrid methods have been proposed, where VLES is one of them. They are all based on the same idea to represent a link between RANS and LES. The hybrid methods aim to combine the computational efficiency of RANS and the potential of LES, where the largest turbulent structures are resolved. In VLES coarser grids and higher Reynolds numbers can be considered. The main difference compared to full LES is that a smaller part of the turbulence spectrum is resolved (Figure 1) and the influence of a larger part of the spectrum has to be predicted by the more advanced model. Commonly in VLES two-equation RANS models are used together with an adaptive filter which can distinguish between the resolved and modeled part of the turbulence spectrum. Thus, VLES can function as anything from RANS to DNS, depending on the filter (Figure 2).

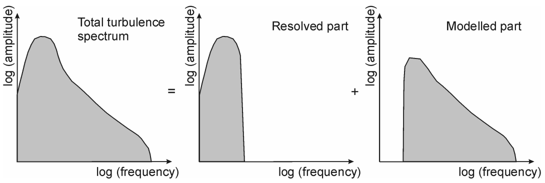

Figure 1: *Modeling approach used in VLES.*

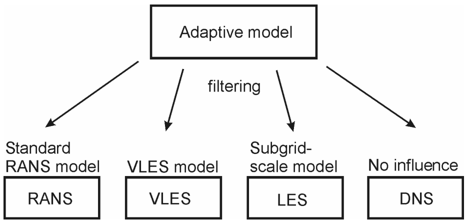

Figure 2: *Principle of filtering and adjustment for adaptive model.*

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 handle unsteady flows better than the standard k-ε model. The extended transport equations for k and ε are given by

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

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

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. In this work applied filtering technique is similar to one by Willems [2]. The smallest resolved length scale Δ used in the filter is, according to Magnato and Gabi [3], dependent on either 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 scales. This means that the filtered dissipation rate must not be influenced be the filter, i.e.

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

However, the turbulent kinetic energy is dominated by the motion of the largest turbulent scales, which potentially can be resolved. The filtered (small scale) turbulent kinetic energy is then obtained by the following expression:

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

As a suitable filter

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

is applied, where

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

contains model constant α, in a range from 1 to 5. It follows that the turbulent length scale L for the whole spectrum is given as

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

The modeled length scales and turbulent viscosity are

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

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

with $c_\mu = 0.09$.

The filtering procedure leads to the final equations

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

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

with the production term

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

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

Another turbulence model which is investigated together with the filtering approach is the modified k-ε model of Reif et al. [5]. This model is essentially the standard k-ε model, where the constant coefficient c_μ is replaced by a function of the strain and rotation rate tensors. The functional form of the coefficient allows the modeled turbulence to relaminarize in the centre of a vortex or in a rotational frame of reference. This quality is a considerable improvement compared to the standard k-ε model, which is known to be insensitive to rotational effects. By allowing the c_μ coefficient to vary, the eddy viscosity decreases in the centre of a vortex core and hence the damping influence of the turbulence model on the unsteady mean flow is less significant.

Numerical methods

FENFLOSS

FENFLOSS (Finite Element based Numerical FLOw Simulation System) is a CFD code based on the finite element method. It is developed at the Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart. It uses 8-node hexahedral elements for spatial domain discretization. The time discretization 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 each element. For flows dominated by advection, a Petrov-Galerkin formulation of 2nd order, with skewed upwind orientated weighting function, is used.

For solving the momentum and continuity equations, a segregated algorithm is used. The equations are linearized 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.

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

The code is parallelized ([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 organized on the level of the matrix-vector multiplication using MIP (Message Passing Interface) on computers with distributed memory and Open MP 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 discretization 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 discretized (linearized) 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 aforementioned models two test cases are chosen. The first test case is swirling flow in a straight pipe, for which experimental data are made available by Steenbergen ([11]). The second test case is a swirling flow through a straight conical diffuser. Experimental data by Clausen ([12]) are used for validation. Both test cases are included in ERCOFTAC database.

Swirling flow in a pipe

The computational domain is a 3D straight pipe with a constant diameter of 0.32 m, in accordance with the experimental setup of Steenbergen ([11]). The available measurements are used for setting the 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 the standard k-ε model, the k-ε of Chen and Kim and VLES, i.e. the filtered version of the Chen and Kim model. As expected the standard k-ε model shows quite poor results in case of intensive swirling flow. The extended model of Chen and Kim shows a less damping characteristic, while VLES manages to resolve clearly unsteady vortex motion. Figure 3 shows a comparison of the pressure fields calculated with the standard k-ε model, the k-ε model of Chen and Kim and VLES.

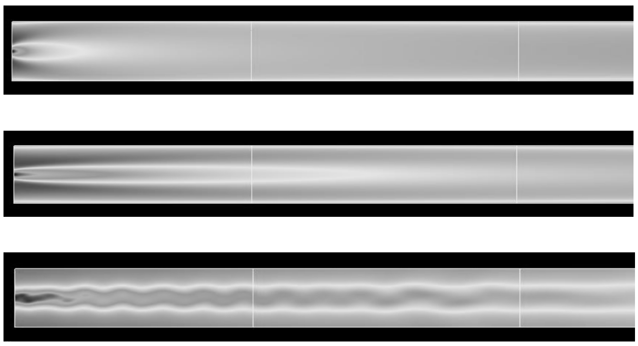

Figure 3: *Pressure filed of unsteady vortex in straight pipe calculated with standard k-ε model (up), extended k-ε model of Chen and Kim (middle) and VLES (down).*

Swirling flow through a straight conical diffuser

CFD code CALC-PMB was used for simulating 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 0.3. 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 modeling small scale turbulence is applied to the standard high Reynolds number (HRN) k-ε model and the k-ε model of Reif et al. The extended model of Chen and Kim is also investigated, but in this case without the filtering approach.

The time-averaged stremwise velocity profiles are shown in Figure 4. The computations correspond well with the experimental data. All models show almost identical time-averaged results, presumably due to the commonly used wall functions and the relatively weak turbulence level in the flow. However, there is a big variety in resolved unsteadiness, visible in the instantaneous solutions. In the simulations using the Chen and Kim model and the filtered standard k-ε model, the solutions near the diffuser outlet are characterized by random turbulence. The solutions obtained using the filtered version of Reif et al. model suggest helicoidal vortex filaments at the same location (Figure 5). Besides, this model seems to allow a higher degree of secondary flow in the diffuser section. The vortices are visualized by isosurfaces of the second invariant of the velocity gradient tensor. The unsteadiness gives rise to pressure oscillations. Fourier transformations of the wall pressure at a point near the diffuser exit are shown in Figure 6. The simulation where the non-filtered standard model is used shows no sign of unsteadiness whatsoever. Only the solutions obtained using the filtered version of the standard model and especially the modified model of Reif et al., show distinct frequencies. In the simulation using the model of Reif et al. the characteristic Strouhal numbers (*St=fD/U*) are 0.22, 0.43 and 0.85. The similar peak at *St*=0.85 is also obtained using the filtered model, whereas the two other peaks are somewhat displaced.

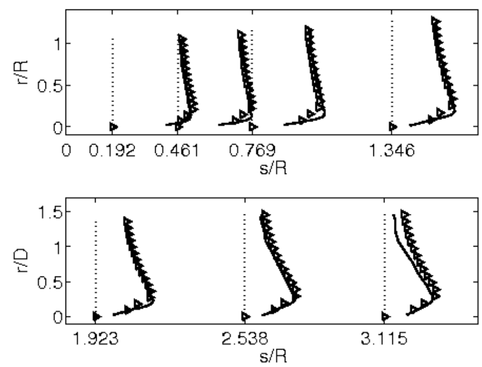

Figure 4: *Streamwise velocity profiles. [Δ]: Calculated, standard k-ε model. [-]: Experiment. The computed results correspond very well to the measured data, and all tested models gives almost identical results for this case.*

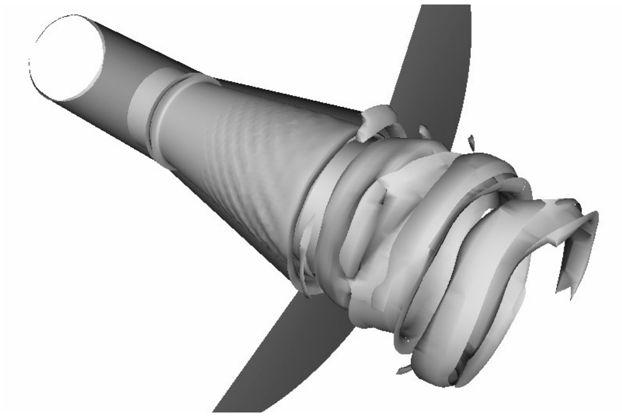

Figure 5: *Isosurfaces of the second invariant of the velocity gradient tensor. Helicoidal vortex filaments are found near the diffuser exit. The filtered turbulence model of Reif et al. has been used.*

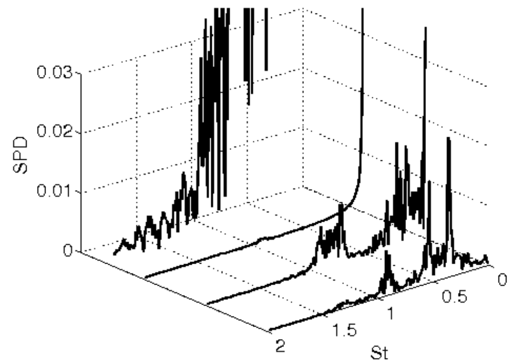

Figure 6: *Spectral power density of the wall pressure near the diffuser exit obtained from different k-ε models on a very coarse grid. Left to right: Extended k-ε model of Chen and Kim; Standard (Jones and Launder) k-ε; Filtered standard k-ε, α=2; Filtered modified k-ε of Reif et al., α=2. The figures are based on 9000 computational time-steps which equals 2.25 s of real time. Only the simulations using the filtered standard model and the filtered modified model of Reif et al. show distinct frequencies.*

CONCLUSIONS

The use of the standard k-ε model leads to rather poor results for unsteady swirling flows. The main reason is that the modeled turbulent eddy-viscosity tends to damp all unsteadiness of the time-resolved flow field. The results can be significantly improved by applying a dynamic filter to the modeled turbulent scales. The filtering procedure yields solutions in which large scale unsteady structures are resolved.

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.