TRENDS AND CHALLENGES IN MODELLING COMPLEX TURBULENT FLOWS

Florian R. MENTER

1 Corresponding Author. ANSYS Germany GmbH. Staudenfeldweg 12, 83624 Otterfing. E-mail: florian.menter@ansys.com

ABSTRACT

The paper gives an overview of trends and challenges in turbulence modelling of complex industrial flows. The emphasis will be on turbomachinery related topics, but will include examples of generic cases and other application areas. The paper will address issues of 2D and 3D separation predictions, curvature effects, laminar-turbulent transition and modelling of unsteady flows.

Keywords: CFD, Turbulence Modelling, SST model, transition, unsteadiness.

NOMENCLATURE

\(\tau_{ij}\) \([Pa]\) Reynolds Stress
\(a_{ij}\) \([Pa]\) Reynolds Stress anisotropy
\(k\) \([m^2/s^2]\) Turbulence kinetic energy
\(\omega\) \([1/s]\) Turbulence Frequency
\(\mu_t\) \([kg/m/s]\) Eddy viscosity
\(S_{ij}\) \([1/s]\) Shear strain tensor
\(T_{ij}\) \([-]\) Tensor base
\(\mu\) \([kg/m/s]\) Dynamic viscosity
\(\rho\) \([kg/m^3]\) Density
\(x_i\) \([m]\) Coordinates

1. INTRODUCTION

Turbomachinery is one of the most challenging industrial application areas for Computational Fluid Dynamics (CFD) methods. One of the reasons for this lies in the many different types of machines with very different geometries and operation principles. In addition, a wide range of physical effects from single phase turbulence and transitional flows to cavitation and complex Fluid Structure Interaction is encountered, depending on machine type and working fluid. Physical modelling is a key element for successful and accurate CFD methods, even though it is acknowledged that numerous effects in blade flow simulations are dominated by the inviscid portion of the equations, like shock losses, tip vortex flows (to a certain extent), secondary pressure driven flows or swirl losses to name the most important. Nevertheless, the accurate modelling of unresolved physics due to turbulence plays a key role, especially in the prediction of off-design conditions, which become increasingly important in modern turbomachinery design.

The current paper will concentrate on the generic aspects of turbulence modelling and their implications for turbomachinery related flows. The author has been involved in a number of projects with an emphasis on turbomachines, but is far from being an expert in this area. The goal is therefore not a comprehensive overview of turbulence modelling for turbomachinery flows, but to lay out the author’s group current approach on addressing some of the challenges encountered in such simulations. The reader is referred to overview articles (Casey, 2002) for a more comprehensive and less subjective picture of the area.

The paper will address the selection of a scale equation as basis for turbulence model development. In addition it will discuss model formulations for flows sensitive to 2D and 3D separation under adverse pressure gradients. Model extensions for swirling and rotating flows will be addressed as well as the importance of including laminar-turbulent transition models into the simulation. Finally, some future directions for including unsteady effects will be outlined.

2. TURBULENCE SCALE-EQUATION

As in any engineer discipline, design decisions have to be made; this is not different in turbulence modelling. One of the most difficult choices is the selection of an optimal (or at least appropriate) scale-equation. Numerous proposals have been made over the last decades, ranging from \(\varepsilon\) to \(\omega\) to \(kL\) and many other variables (see e.g. Wilcox 1998). There is very limited theoretical justification for choosing one over the other (an exception would possibly be the \(kL\) and related equations, as they can be based on an exact transport equation, Rotta,
The selection is therefore based on practical arguments and long term experience in using these equations. The author’s choice has for many years been the $\omega$-equation for the following reasons:

- Robust behavior even in complex geometries and on non-optimal grids.
- Simple and robust Viscous Sublayer Model (VSM) formulation (simple $y^+$-insensitive formulation possible).
- Accurate near wall behavior for adverse pressure gradient flows (Wilcox 1998, Menter, 1994).
- Ability to combine with engineering transition models Menter et al (1996).
- Accurate heat transfer prediction capabilities (Esch and Menter, 2003).

The starting point for many of the authors’ model formulations and extensions is the BSL $\omega$-equation (Menter, 1994, based on a blending of the $k$-$\omega$ model and the $k$-$\varepsilon$ model). The BSL model avoids the freestream sensitivity of the Wilcox model (Menter, 1994, Menter, 2009, Wilcox, 1998) while maintaining the desirable characteristics of that model in the near wall layer. The price for the blending is the need for computing blending functions based on the wall distance. This is however not a significant cost, as the wall distance can be obtained from the solution of a Poisson equation (for fixed geometries only at the start of the simulation). In addition, the wall distance is needed also for numerous other models – most prominently transition models, and has to be computed anyway.

Figure 1 shows the way that different modelling elements can be grouped around the BSL $\omega$-equation. It is important to ensure that all (reasonable) combinations of models can operate in combination.

![Figure 1: Grouping of model extensions around the BSL $\omega$-equation](image)

The BSL models attraction results from the accurate and robust performance of the underlying Wilcox model in the inner part of the turbulent boundary layer (sub- and logarithmic layer). The Wilcox model does not require non-linear damping terms in the viscous sublayer due to its boundary condition for $\omega$. It thereby avoids many of the numerical problems encountered when integrating the $\varepsilon$-equation to the wall. In a recent overview article (Menter, 2009), the current author made the argument, that the $\omega$-equation is a very effective elliptic relaxation model as it reduces near the wall to:

$$\frac{\mu}{\partial x_j} \frac{\partial^2 \omega}{\partial x_j^2} = \beta \rho \omega^2$$

which together with the boundary condition for $\omega$ transports information on the wall presence into the near wall layer. In that respect, the model is substantially simpler than the $\varepsilon$-equation-based elliptic relaxation formulation proposed by Durbin (1991), while serving essentially the same purpose.

The ability of turbulence models to faithfully represent the boundary layer depends just as much on the resolution of the numerical grid as on the accuracy of the turbulence model. The author would argue strongly in favor of a full resolution of the boundary layer including the viscous sublayer – meaning that grids with $y^+ < 1$ should be applied routinely. Especially for turbomachinery flows, which are often characterized by small to moderate Reynolds numbers, the use of wall functions introduces an unacceptably large source of error into the simulations. Considering the computing power available today, the additional number of nodes required for the resolution of the viscous sublayer is easily achievable and leads to a substantial reduction of computational uncertainties. It is helpful that there is a strong drive for including effects of laminar-turbulent transition in today’s CFD simulations (see below) which automatically results in highly resolved grids, due to the need of resolving the thin laminar boundary layer upstream of transition.

Another industrial standard developing over the years is the formulation of wall boundary conditions which are $y^+$-insensitive, meaning that over a wide range of $y$ values, a constant wall-shear and wall-heat transfer coefficient can be maintained. It is certainly the goal in the author’s group to avoid $y^+$-dependencies for all models implemented into our CFD codes. Still, even with $y^+$-insensitive formulations, $y^+ < 1$ is desirable.

### 3. STRESS-STRAIN RELATIONS

#### 3.1. SST Limiter

Most engineering simulations are based on Eddy-Viscosity Models (EVM), resulting in a linear tensorial dependence of the Reynolds Stress tensor, $\tau_{ij}$, on the tensor of the shear strain, $S_{ij}$.
\[ \tau_{ij} = \mu \partial_{ij} \frac{2}{3} \rho k \delta_{ij} \]  

(2)

While the tensorial form is identical for all linear EVM, the formulation of the eddy-viscosity has a substantial effect on the accuracy of the simulations. One example is the modification used in the Shear Stress Transport (SST) model (Menter, 1994), where the standard formulation \( v' = k \omega \) is replaced by:

\[
   v' = \frac{a_k}{\max(a_k, \omega, S F_1)} \quad S = \sqrt{2 S_y S_y} \]

(3)

where \( a_k = 0.31 \), \( k \) is the turbulence kinetic energy, \( \omega \) is the turbulence frequency, \( S \) is the shear strain rate, and \( F_1 \) is a blending function activating the limiter only in boundary layer flows. This formulation enforces the relation:

\[
   \tau_{ij} = -a_k \rho k \]

(4)

as indicated by experiments. This small change in the definition of the eddy-viscosity has resulted in a major improvement of the SST models ability to predict adverse pressure gradient flows. Figure 2 shows a comparison of the SST model with other modern eddy-viscosity models for the flow around the NACA 4412 airfoil of Coles and Wadcock (1979).

![Figure 2: Results of the modern eddy-viscosity models for the flow around a NACA 4412 airfoil (\( \alpha = 13.87^\circ \), \( \text{Re} = 1.5 \times 10^6 \)). Velocity profiles at different stations.](image)

In some turbomachinery applications, especially axial compressors, there is an indication of overly large separation zones predicted with the SST model. There is a tendency to therefore turn back to the \( k-\epsilon \) or the Wilcox \( k-\omega \) model. However, the former has the disadvantage (amongst others) of not being compatible with transition modelling, while the latter introduces significant freestream sensitivity into the simulations. The reasons for premature stall and larger separation zones are most likely not related to an early separation prediction, but an overprediction of the length of the separation bubble. Switching to models which are known to delay separation for 2D validation cases, results most likely in a cancellation of errors. Nevertheless, it might still be desirable from an overall design standpoint to balance errors to improve overall agreement with data. It is therefore important to note that the SST model parameter \( a_1 \) can be increased (from 0.31) to reduce separation tendencies. This is not generally recommended, but would still be the preferred option, compared to switching to the \( k-\epsilon \) or the \( k-\omega \) model for the above reasons.

### 3.1. EARSM

Explicit Algebraic Reynolds Stress Models (EARSM) offer an attractive intermediate solution method between EVM and full RSM. Numerous variants of such formulations have been proposed (e.g. Gatski and Speziale, 1993, Pope, 1975, Rodi, 1976, Wallin and Johansson, 2000, Hellsten and Laine, 2000). The author’s group (Menter et al.2009) has recently investigated several adaptations of the Wallin-Johansson (2000) EARSM. The goal of the investigation was to improve the predictions of flows separating from a corner under an adverse pressure gradient. Such situations are observed in wing-body – or blade-hub-shroud regions. It is found that EVM have a tendency of predicting the separation point in such regions too far upstream, resulting in a strong overprediction of the corner separation zone. In some cases, it was even observed that EVM produce a different (and incorrect) flow topology. In Menter et al. (2009) it is argued that the main effect missing in EVM in this situation is the anisotropy of the normal stresses, which drives a secondary flow into the corner. This secondary flow energizes the boundary layer near the corner walls and prevents (delays) separation. His effect can in principle not be captured by linear EVM and justifies the increased complexity of EARSM.

It is beyond the scope of the present article to present the full formulation of the EARSM which is given in Wallin and Johansson (2000) and Menter et al. (2009). It is sufficient to show the main equation relating the stress and the strain tensors:

\[
   a_{ij} = \beta_1 T_{1i,j} + \beta_2 T_{2i,j} + \beta_3 T_{3i,j} + \beta_4 T_{4i,j} + \beta_5 T_{5i,j} + \beta_6 T_{6i,j} + \beta_7 T_{7i,j} + \beta_8 T_{8i,j} + \beta_9 T_{9i,j} + \beta_{10} T_{10i,j} \]

(5)

where \( a_{ij} \) is the anisotropy tensor, \( T_{ki,j} \) are a tensor base formed by the strain and vorticity tensor and the \( \beta_i \) are coefficients determined from an underlying RSM. It should further be noted that the models proposed in Menter et al. (2009) are combined again with the BSL model. The
coefficient $\beta_1$ has a very close similarity to the formulation of the SST EVM and thereby largely preserves the separation prediction capability of the SST model for 2D flows.

Figure 3 shows the flow computed with the EARSM for a rectangular channel with corresponding velocity components in streamwise ($x$) and diagonal direction ($y=z$) along the channel diagonal. The important effect is the increase (relative to the EVM) of the axial velocity in the near wall region of the corner, caused by the secondary flow into the corner.

Figure 3: Comparison mean axial (left) and diametric velocity along line $y=z$ (right) with DNS of Huser and Biringen (1993).

Figure 4 shows a testcase with a drastic effect of the change in the stress-strain relationship. It is the 3D diffuser flow of Cherry et al. (2007). In this flow, the full EARSM, the linear version of the EARSM (only $\beta_1$ active) and the SST models are compared against exp. data. It can be seen in Figure 5 that the linear version of the EARSM and the SST model produce an incorrect flow topology with the separation on the wrong side of the diffuser.

Figure 4: Computational domain and grid used for the rectangular diffuser flow case

Figure 5: Comparison velocity fields at section $x=16$ cm (last section shown in Figure 4).

Figure 6 shows the pressure distribution along the lower channel wall. Clearly the incorrect flow topology results in a significantly lower pressure rise and a strong discrepancy with the experiments. It should be noted that this testcase is an extreme example reacting with a change in flow topology to model differences. Such drastic differences between models are not expected in most industrial flows.

This example is however relevant for some turbomachinery applications where one has to deal with low-aspect-ratio channels (radial compressors) close to separation. In Menter et al. (2009) another example is given for an airplane wing-body junction, where the separation in the wing-body corner is significantly reduced and in better agreement with the oil-flow data for the EARSM. This is again of relevance to turbomachinery flows, where the blade aspect ratios are often small and excessive corner separation prediction can significantly alter the blade performance.

Figure 6: Comparison of pressure coefficient for 3D Diffuser on lower wall

3. STREAMLINE CURVATURE AND SYSTEM ROTATION

By the nature of the application, many turbomachinery flows are affected by effects of streamline curvature and system rotation. This is not only the case for blade row simulations, but also for auxiliary flows in draft tubes or swirl-stabilized
flows in combustion chambers. Again, the effect cannot be captured by EVM without additional enhancements. Even EARSM do not include the effects in its entirety, unless additional curvature correction (CC) terms are introduced.

In the framework of the SST model, a CC proposed by Spalart and Shur (1997) has been slightly modified and adapted to the requirements in two-equation models (Smirnov and Menter, 2008).

Figure 7: Developed channel flow at Re = 5800; comparison with DNS of Kristoffersen and Andersson.

Figure 7 shows velocity profiles computed in a rotating channel flow at different rotation rates. The dashed line is the SST model without the CC correction and the full line the modified version of the model. The SST-CC model accurately represents the effect of the system rotation on the flows, which cannot be captured without such amendments.

Figure 8 shows a hydro-cyclone as studied experimentally by Hartley (1994). This flow is dominated by a strong swirl inside similar to flows in combustion chambers or draft tubes. Figure 9 shows tangential velocity profiles in a hydro-cyclone. Experimental data are available for the axial and tangential velocity components at five vertical sections shown in Figure 9.

Figure 8: Schematic of the hydro cyclone, computational domain, and positions of measurement planes (experiments of Hartley, 1994).

Figure 9: Time-averaged profiles of the tangential velocity in the hydrocyclone; comparison with experiments of Hartley (1994).

Another interesting application is for tip vortex flows shown in Figure 10. A generic testcase is the wing tip flow studied experimentally by Chow et al. (1997). The wing has a rounded tip and is placed in the wind tunnel at an angle of attack $\alpha = 10^\circ$. The chord-based Reynolds number is 4.6 million and the Mach number is around 0.1.

Figure 10: Computational domain and grid used for NACA 0012 wing with rounded tip (experiments of Chow et al., 1997).

The flow was simulated with a grid off $4.6 \times 10^6$ nodes. Figure 11 shows the axial velocity profiles at the X/C=0.24 downstream station. It is clear that the CC modification improves the agreement with the...
data, albeit not to perfection (see Smirnov and Menter, 2008). However, the integrity of the vortex is preserved longer than without CC, which is a result of the reduced eddy-viscosity inside the vortex.

4. LAMINAR-TURBULENT TRANSITION

Modelling of laminar-turbulent transition in boundary layers has proven one of the most challenging tasks in CFD for many decades. While many industrial flows are in the range of $10^5 < \text{Re} < 10^6$, meaning in regimes where significant portions of the boundary layers can be laminar, there was simply no reliable way of including these effects even to first order in general-purpose CFD codes. A model extension to the SST model which allows the inclusion of such effects into general CFD simulations has recently been developed by the authors’ group (Menter et al, 2006, Langtry et al, 2006). The model solves two additional transport equations and incorporates experimental correlations to trigger the transition onset. The model formulation is strictly local and therefore fully compatible with modern general purpose CFD codes. Discussion of the model equations is beyond the scope of the current article and is therefore not presented here.

The transition model has been validated against a wide range of experimental data and is used today successfully in many industrial CFD simulations. The application range covers turbomachinery blades, wind turbines, racing cars all the way to the design of sailing yachts for the Americas cup.

Huang et al. (2003) conducted experiments on the PAK-B blade cascade for a range of Reynolds numbers and turbulence intensities. The experiments were performed at the design incidence angle for Reynolds numbers of 50,000, 75,000, and 100,000 based on inlet velocity and axial chord length, with turbulence intensities of 0.08%, 2.35% and 6.0% (which corresponded to values of 0.08%, 1.6%, and 2.85% at the leading edge of the blade).

The computed pressure coefficient distributions for various Reynolds numbers and freestream turbulence intensities compared to experimental data are shown in Figure 12. In this figure, the comparisons are organized such that the horizontal axis denotes the Reynolds number whereas the vertical axis corresponds to the freestream turbulence intensity of the specific case. As previously pointed out, the most important feature of this testcase is the extent of the separation bubble, characterized by the plateau in the pressure distribution. The size of the separation bubble is actually a complex function of the Reynolds number and the freestream turbulence value. As the Reynolds number or freestream turbulence decrease, the size of the separation and hence the pressure plateau increases. The computations with the transition model compare well with the experimental data for all of the cases considered, illustrating the ability of the model to capture the effects of Reynolds number and turbulence intensity variations on the size of a laminar separation bubble and the subsequent turbulent reattachment.

Other approaches to transition modeling have recently become available. One of the more promising is from Walters and Cokljat (2008). It would however be desirable to combine this formulation with the BSL/SST model in order to avoid the freestream sensitivity of the standard $k-\omega$ model underlying its current formulation.

It is important to note that transition models can only be combined with turbulence models which do not mimic transitional behavior due to their VSM formulation. Otherwise, the interference of both formulations would result in an unpredictable behavior.

5. UNSTEADY FLOW SIMULATIONS

The SST model is currently linked to two scale-resolving methodologies, namely the Detached Eddy Simulation (DES) of Spalart (2000), and Sterelets (2001) and the Scale-Adaptive Simulation (SAS) model (see e.g. Menter and Egorov (2009)). The goal of both formulations is to cover the wall-boundary layers with RANS while allowing the formation of turbulent eddies in large separation zones. In DES this is achieved by forcing the turbulence model into an LES formulation if the grid spacing is smaller than a constant times the turbulent length scale. The SAS model relies on the...
introduction of the von Karman length scale, which allows the model to adjust to resolved scales.

The derivation of the SAS model is based on an exact transport equation for the integral length-scale as derived by Rotta (1972). It was shown that one of the leading order terms in this equation should contain the second spatial derivative of the velocity field. This results in the introduction of the von Karman length scale into the equations:

$$L_{vk} = \kappa \left| \frac{\partial^2 U}{\partial y^2} \right|$$

(6)

(in a general purpose code, the derivatives are computed from coordinate-independent formulations). The introduction of $L_{vk}$ enables the model to detect already resolved structures in the flow field and thereby allows the evolution of a turbulent spectrum in unstable flow regions. This is illustrated in the right side of Figure 13 which shows the SST-SAS model, which has been augmented by $L_{vk}$.

An interesting application of the SST-SAS models is for the film cooling of the trailing edge of a turbine blade. Due to the decreasing thickness of the blade near the trailing edge, the cooling is achieved by a cooling film injected parallel to the blade surface. Figure 14 shows the geometry of the blade and the surface grid for the AITEB testcase by Martini et al. (2003). There are two inlet regions for this simulation. On the upper inlet, the hot gas enters the domain and at the inlet to the cooling channel, cold gas is injected. The cold gas does however pass over a hot wall before it reaches the mixing zone. It does therefore not stay at the inlet temperature. The reference temperature for the cold gas is taken downstream of the cold gas inlet. It is therefore not the value of the cold gas at the inlet due to heat transfer from the walls. In the simulations, the reference temperature was taken at the same location as in the experiment. The upper boundary of the domain is a free slip adiabatic wall. Periodicity is applied at the side planes. This testcase is courtesy of Dr. Lutum of MTU Aero Engines and has been investigated within the EU-project AITEB, G4RD-CT-1999-00055.

The finite-volume grid consists of $6.48 \times 10^5$ hexahedral elements. Most of the walls are resolved with the $y^+$ values below 1. A time step of 0.01 ms was used. (Typical velocity 50 m/s and dimension $L \sim 0.1$m)

The main parameter for the evaluation of the device is the cooling efficiency. It is defined as follows:

$$\eta = \frac{T_{in}^{hot} - T_w}{T_{in}^{hot} - T_{ref}^{cold}}$$

(7)

where $T_{in}^{hot}$ is the temperature of the hot gas at the inlet, $T_w$ is the computed wall temperature at the adiabatic wall section, and $T_{ref}^{cold}$ is the reference low temperature taken at the reference point shown in Figure 14. Unfortunately, there is an uncertainty in the reference temperature, which has resulted in a relative shift of the experimental and computed results. Despite this shift, which is observed for all turbulence models, clear differences in model performance can be discerned.

Figure 15 shows the cooling efficiency (averaged in time and spanwise direction) for three different simulations against the experimental data. Clearly, both steady RANS and URANS do not provide sufficient mixing to reproduce the experimental results. The cooling efficiency is therefore computed too optimistic, as the trailing edge surface is shielded from the hot gas. The SAS model produces a significantly stronger mixing of the two streams and results in a much better agreement of the cooling efficiency with the experiments.
Figure 15: Cooling efficiency for different versions of the SST model compared to experimental data

Figure 16 shows the turbulent structures computed by the SAS model. They represent an iso-
surface of $Ω^2-S^2=10^5$ 1/s². It shows the intensive mixing by turbulent structures in the bluff body region behind the separator of hot and cold air.

Figure 16: Turbulent structures computed by the SST-SAS model for a film cooling test case

Another interesting application area for the SAS model in turbomachinery flows is for rotating cavities in gas turbines. Figure 17 shows the resolved scales in such a simulation carried out for a high-pressure turbine (HTP) (see Smirnov et al. 2009). Figure 17 shows the turbulent structures resolved inside the cavity.

The details of the simulation can be found in Smirnov et al (2009) (including comparisons with simplified reference cases). It suffices to say here that a significant improvement relative to RANS simulations could be achieved in the main quantity (the temperature rise from inlet to outlet). While RANS predicted virtually no increase in temperature, the SAS resulted in an increase of 29°C, which was well within the measurement data scatter (36 ± 11 °C).

Figure 17: Isosurfaces of the Q invariant (colored by the eddy viscosity ratio) computed on the refined grid.

6. SUMMARY

An approach for turbulence modeling for turbomachinery flows has been outlined. It consists of:

- $ω$-based BSL scale equation
- advanced stress-strain relationships (SST, EARSM)
- enhancements for streamline curvature and system rotation
- model equations for transition prediction
- enhancements for unsteady flow simulations

The rationale behind these choices has been outlined. For each area, testcase results for generic and turbomachinery related simulations have been given.

The author is confident that the current approach provides the flexibility of including the most relevant physical effects in a robust and accurate manner into CFD simulations of turbomachinery flows. In combination with DES and SAS model extensions, the framework should also provide an attractive pathway into unsteady simulations. In addition, the approach allows the combination of all elements in a single simulation.

ACKNOWLEDGEMENTS

The author wants to thank his colleagues, Richard Lechner, Yuri Egorov and Davor Cokljat for providing the code infrastructure and model implementation, as well as computing some of the testcases. In addition the fruitful cooperation with NTS in St. Petersburg is acknowledged, especially the contributions of Pavel Smirnov and Andrey Garbaruk.
REFERENCES


