Performance of RANS turbulence models for the numerical simulation of the flow affected by micro vortex generators

Additional Thesis

Committee
Prof.dr.ir. W.S.J. Uijttewaal
Dr.ir. A.H. van Zuijlen
Ir. L. Florentie

Student
P.C. Meijers, BSc
14 December 2015
Contents

1 Introduction 3
   1.1 Objective and approach .............................................. 5
   1.2 Structure of this report .............................................. 5

2 Vortex generators 6
   2.1 The turbulent boundary layer ....................................... 6
   2.2 Flow separation ...................................................... 8
   2.3 Types of vortex generators ......................................... 10
   2.4 Experimental and numerical research ............................... 12

3 Test case setup 13
   3.1 Experimental setup .................................................. 13
   3.2 Numerical setup ...................................................... 14
   3.3 Turbulence models ................................................... 16
      3.3.1 Standard \( k-\varepsilon \) .......................................... 16
      3.3.2 \( k-\omega \)-SST .................................................. 17
      3.3.3 Launder Gibson RSTM .......................................... 17
      3.3.4 Speziale Sarkar Gatski RSTM .................................. 17

4 Results 18
   4.1 Mean flow ............................................................ 18
   4.2 Turbulent quantities ................................................ 21

5 Conclusion and recommendations 28
   5.1 Conclusion ........................................................... 28
   5.2 Recommendations .................................................... 29
Bibliography

A Turbulence models

A.1 Standard $k-\varepsilon$ ......................................................... 32
A.2 $k-\omega$-SST ................................................................. 32
A.3 Launder Gibson RSTM ...................................................... 33
A.4 Speziale Sarkar Gatski RSTM ........................................... 34
Chapter 1

Introduction

Over the past decades the demand for wind energy has seen a rapid increase. This demand has lead to the development of larger and more efficient wind turbines. Most of these new wind turbines are placed in large wind farms. Wind farms are often built offshore, because of the presence of more persistent wind and the smaller visual impact compared to land based wind parks. In Figure 1.1 an example of an offshore wind farm is shown.

![Offshore wind farm off the coast of the United Kingdom.](image)

**Figure 1.1** Offshore wind farm off the coast of the United Kingdom.

The design and construction of offshore wind turbines requires expertise from different engineering fields. The construction, stability and strength of the tower are challenges that are addressed in the fields of offshore and structural engineering, two branches of civil engineering. The design of the turbine blades is a research topic within aerospace engineering. It can be said that offshore wind energy is a multidisciplinary research field.

The aerodynamic properties of the turbine blades determine to a large extend the energy output of the wind turbine. A small alteration in the design of the blades can result in different lift and drag properties. These properties are influenced by phenomena like flow...
separation, where the flow over the blade does not stay attached to the surface of the blade. This results in extra drag or even stall when the separation becomes large. When the blade experiences stall the energy output is reduced to a minimum. It is therefore critical to delay or even prevent the flow from separating.

A way to delay the separation is the use of vortex generators. These are small structures on the blade that disturb the flow field creating a streamwise vortex. An example of a wind turbine blade equipped with vortex generators is shown in Figure 1.2. The vortex transports more energetic fluid from the freestream towards the blade. The slower moving fluid close to the blade is energised by this process, moving the point of separation further downstream.

![Figure 1.2](image)

**Figure 1.2** Vortex generators on a wind turbine. One vortex generator pair is indicated with the arrow.

A vortex generator produces drag of its own because it disturbs the flow field. In an effort to reduce this extra drag smaller vortex generators have been developed: the so called micro vortex generators. These devices produce a weaker vortex so the positive effect of the vortex generators is confined to a smaller distance downstream. This has increased the need to accurately model the effects of the vortex generator.

The effects the vortex generators have on the flow can be addressed numerically by means of a Reynolds Averaged Simulation. This approach is the least expensive way to simulate the turbulent flow over a vortex generator in terms of computation time. This low computational cost has the advantage the method can be used during a design optimisation process, in which many different designs have to be compared.

Within a RANS simulation the effects of the turbulence are modelled by turbulence models. Many of these models have been developed over the years, each within their own field of application. The models are often tuned to match simple experimental results, therefore it is not clear if the models will correctly predict the flow field if more complex flow phenomena are present. The outcome of a RANS simulation should always be validated with an experiment.
1.1 Objective and approach

The use of micro vortex generators to improve the flow over wind turbine blades has increased the need for accurate methods to simulate the flow. This implies that the used RANS turbulence model should be capable of correctly capturing the effects of the vortices. The objective of this thesis is therefore

Assessment of the performance of different RANS turbulence models in predicting the flow field and the turbulent quantities created by a micro vortex generator.

This objective gives rise to the following research questions:

- How do vortex generators affect the flow over an airfoil?
- What types of vortex generators are available and what are their characteristics?
- What is the influence of the vortex on the turbulent boundary layer?
- What research has already been done regarding RANS models and micro vortex generators?
- How well are the mean flow field and the turbulent quantities predicted by the RANS models?

These questions will be answered by a literature review about the experimental and numerical research. After this review the flow field over a micro vortex generator pair will be simulated using different RANS turbulence models. The results from these simulations are then compared to the results of an experiment performed on the same geometry.

1.2 Structure of this report

The second chapter of this thesis is a short discussion of vortex generators and why they are used to delay flow separation. In Chapter 3 the setup of the experimental and numerical case is presented. The used turbulence models are also presented in that chapter. The results from the experiment and the simulation are presented and discussed in Chapter 4. The last chapter gives the conclusion and recommendations for further research. In the appendix the turbulence models are shortly described.
Chapter 2

Vortex generators

To understand why vortex generators are used to improve wind turbine performance, the problem of flow separation needs to be explained. This will be done in the first part of this chapter, after the boundary layer has been introduced. Different types of vortex generators developed and tested over the years will be discussed after that. The last part of this chapter contains a short overview of the experimental and numerical research that has been performed and published in literature.

2.1 The turbulent boundary layer

When a fluid flows over a solid surface, for example a plate, a boundary layer is formed. This is a result of the no slip condition stating that at the wall the velocity $u$ is equal to zero. Within in the boundary layer this velocity component varies from zero at the wall to $U_\infty$ at the edge of layer. In Figure 2.1 the velocity profile in the boundary layer is shown. The figure also shows a definition of the boundary layer thickness $\delta$, which can be defined as the distance normal the surface where the velocity is equal to $0.99U_\infty$.

![Figure 2.1 Streamwise velocity profiles in a boundary layer. The figure is based on [Dowling et al., 2012].](image)

Within the boundary layer the motion of the fluid can be described by the steady state boundary layer equations. These equations are derived from the Navier-Stokes equation in
These equations will be used in the next paragraph to investigate the process of flow separation. Before the separation is discussed, further insight in the structure of a turbulent boundary layer will be given.

As the boundary layer develops downstream it thickens. At some point the laminar boundary layer becomes unstable and transitions into a turbulent boundary layer. The flow becomes fully three dimensional and time dependent. A common way to assess this turbulent flow is to look at the time averaged flow. The instantaneous velocity field is split into a mean and a fluctuating part, the so called Reynolds decomposition. When the Navier-Stokes equations are time averaged and decomposed according to this decomposition the Reynolds Averaged Navier-Stokes equations are obtained.

\[
\frac{\partial U_i}{\partial x_i} = 0
\]  

(2.4a)

\[
U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial U_i}{\partial x_j} - \langle u_i u_j \rangle \right). 
\]  

(2.4b)

The influence of the turbulent fluctuations on the mean flow is all contained in the symmetric Reynolds stress tensor \( \langle u_i u_j \rangle \).

When the flow quantities within a turbulent boundary layer are averaged over time the flow can be regarded as a statistically two dimensional flow. In the time averaged flow only one mean velocity gradient, \( \frac{\partial U_i}{\partial y} \), is unequal to zero. Figure 2.2 shows the profiles of the Reynolds stresses \( \langle u_i u_j \rangle \) across the boundary layer.

**Figure 2.2** Profiles of Reynolds stress. (a) Across a turbulent boundary layer. (b) In the viscous wall region. Taken from [Pope, 2000].

To get a further insight in the turbulent structure of a turbulent boundary layer the anisotropy of the Reynolds stresses is considered. This is an important turbulent quantity, because only
the deviatoric or anisotropic part of the Reynolds stress tensor is successful in transporting momentum. [Pope, 2000] The Reynolds stress anisotropy tensor is defined as the normalised deviatoric part of the Reynolds stress tensor:

\[
b_{ij} = \frac{\langle u_i u_j \rangle}{2k} - \frac{1}{3} \delta_{ij}, \tag{2.5}\]

where \( k = \frac{1}{2} \langle u_i u_i \rangle \) is the turbulent kinetic energy and \( \delta_{ij} \) is the Kronecker delta. To gain insight in the properties of this tensor and thus in the anisotropy of the Reynolds stresses it is convenient to look at the invariants of the tensor. The invariants of a second order tensor are given by

\[
I_b = b_{ii} = \text{tr} (b_{ij}) \\
II_b = b_{ij} b_{ji} = \|b_{ij}\|^2 \\
III_b = b_{ij} b_{jk} b_{ki}.
\]

The first invariant of \( b_{ij} \), \( I_b \), is zero due to the definition 2.5. The other two invariants can be used to characterise the state of turbulence. [Pope, 2000] For example for isotropic turbulence the second and third invariant of \( b_{ij} \) are equal to zero. In the freestream, outside the boundary layer, the turbulence is isotropic. Within the boundary layer the anisotropy increases when the wall is approached, something that can be seen from the Reynolds stress profiles in Figure 2.2. In Figure 2.3 the two invariants of the undisturbed turbulent boundary layer as measured in the experiment of [Baldacchino et al., 2015] are shown, displaying the same increase in anisotropy as the wall is approached.

![Figure 2.3](image1)

(a) \( II_b \)  
(b) \( III_b \)

**Figure 2.3** Invariants of the Reynolds stress anisotropy tensor \( b_{ij} \) in an undisturbed boundary layer. Calculated with the data from [Baldacchino et al., 2015].

### 2.2 Flow separation

Now the boundary layer has been introduced, separation can be addressed. The blade of a wind turbine is shaped like an airfoil, which is a curved surface. The shape of the surface
determines the resulting flow field. The flow field over a general curved surface is visualised by means of streamlines and shown in Figure 2.4. The $x$-axis is defined in streamwise direction. The $y$-axis is defined normal to the wall.

The shape of the velocity profiles in Figure 9.12 suggests that a decelerating exterior flow tends to increase the thickness of the boundary layer. This can also be seen from the continuity equation:

$$
\rho \frac{\partial}{\partial y} \left( \frac{\partial u}{\partial y} \right) = - \int \frac{du}{dx} \, dy.
$$

Compared to flow over a flat plate, a decelerating external stream causes a larger $-\frac{du}{dx}$ within the boundary layer because the deceleration of the outer flow adds to the viscous term. The surface shear stress and stream-wise fluid velocity near the surface are highest and lowest in the favorable and adverse pressure gradients, respectively, with the $dp/dx = 0$ case falling between these limits.

When air flows over the surface a turbulent boundary layer develops along the wall. At the wall the steady state boundary layer equation, equation 2.3, reduces to

$$
\mu \left( \frac{\partial^2 u}{\partial y^2} \right)_{wall} = \frac{dp}{dx}.
$$

From the velocity profiles in Figure 2.4 the sign of the second derivative $\frac{\partial^2 u}{\partial y^2}$ at the top of the boundary layer is shown to be negative in order to match the free stream velocity in a smooth way. The sign of the second derivative at $y = \delta$ will not change further downstream, in contrast to the sign of $\left( \frac{\partial^2 u}{\partial y^2} \right)_{wall}$.

At the start of the airfoil the flow is accelerated and the pressure gradient $\frac{dp}{dx} < 0$. The pressure gradient is then called favourable. From equation 2.6 it is clear that in this case $\left( \frac{\partial^2 u}{\partial y^2} \right)_{wall}$ is also negative. The second derivative stays negative in the whole boundary layer. This is no longer the case when the fluid is decelerated in the second part of the airfoil and the pressure gradient is positive, $\frac{dp}{dx} > 0$. This is called an adverse pressure gradient. Equation 2.6 now implies that $\left( \frac{\partial^2 u}{\partial y^2} \right)_{wall}$ is also positive. As a consequence the sign of the second derivative changes somewhere in the boundary layer, meaning that the streamwise velocity $u$ has a point of inflection. This is an important fact for boundary layer stability. [Dowling et al., 2012]

When the adverse pressure gradient is strong or acts over a long enough distance, the flow direction at the wall reverses. The point where the wall shear stress $\mu \frac{du}{dy} = 0$ is called the separation point. The streamlines are diverted away from the wall and a region of recirculating fluid is created. This region results in an increase of the drag experienced by the object. Separation should occur as far downstream as possible in order to minimise the drag experienced by the airfoil, hence increasing the performance of the turbine.

The position of the separation point depends on the shape of the airfoil and on the Reynolds number. In a low Reynolds number (laminar) flow separation occurs much earlier than in a high Reynolds number (turbulent) flow. This is caused by the velocity profile in the boundary.
The difference in these profiles is shown in Figure 2.5. The turbulent boundary layer has more high speed fluid closer to the wall and a higher wall shear stress, resulting in a delay of the separation.

To further delay the separation Taylor proposed the use of vortex generators in the late 1940s. [Schubauer and Spangenberg, 1960] The vortex generators create a streamwise (longitudinal) vortex in the boundary layer, locally enhancing the streamwise velocity profile by transporting high momentum fluid from the freestream towards the wall. The new profile is also shown in Figure 2.5, where it can be seen to be fuller than an undisturbed boundary layer. This way even larger adverse pressure gradients can be encountered without separation. The effectiveness of a vortex generator in enhancing the velocity profile is dependent on the shape and size of the device. These aspects will be discussed next.

2.3 Types of vortex generators

A vortex generator is a small structure that disturbs the flow field by creating a streamwise vortex. The shape of these small structures partly determines their effectiveness in mixing the high momentum fluid with the low momentum fluid and the amount of the drag the device produces. [Schubauer and Spangenberg, 1960] They concluded that the most effective designs are vanes and wedges, because they produce the most energetic vortices and suffer from less drag than other designs. These types of vortex generators are shown in Figure 2.6.

A further classification can be made based on the height of the vortex generator compared to the local boundary layer thickness. The height of conventional vortex generators is in the same range as the boundary layer thickness. If the height is smaller than 10 to 50% of the local boundary layer thickness the vortex generator is called a sub boundary layer vortex generator, micro vortex generator or low profile vortex generator. [Lin, 2002]. Micro vortex generators were developed to reduce the extra drag produced by conventional vortex generators. The smaller size of the devices does reduce the drag, but the vortices that are generated are weaker.
Figure 2.6  Wedge vortex generator (left) and vane vortex generator (right)

than the ones produced by the conventional devices. The weaker vortices are still effective in delaying the separation, [Fernández-Gámez et al., 2014], however the position of the devices on the airfoil becomes more important for micro vortex generators. The reason for this is that the weaker vortices dissolve much quicker downstream, thereby making the positive effect of the vortex generator more local. This means that correctly modelling the flow behind the device is very important for an airfoil design with micro vortex generators.

A third distinction is possible based on the rotational direction of the vortices that are generated when the devices are placed in an array. When the vortices rotate in the same direction the vortices are called co-rotating. When the rotation direction changes between each vortex, the vortices are called counter-rotating. The difference is shown in Figure 2.7. Co-rotating vortices lose their strength more quickly than counter-rotating vortices, because the co-rotating vortices merge which causes a significant loss in vortex strength. [Pauley and Eaton, 1988] For counter-rotating vortices a further distinction can be made based on the flow direction between a vortex pair. If the flow is towards the wall the flow is categorised as common flow down. For a common flow up vortex pair the flow between the vortices is away from the wall.

Figure 2.7  Co-rotating vortex pair (left), common flow up vortex pair (middle) and common flow down vortex pair (right)
2.4 Experimental and numerical research

Conventional vortex generators have been tested over the years by several researchers. The effectiveness of different shapes of vortex generators has been tested experimentally by [Schubauer and Spangenberg, 1960], where it was shown that the vane and wedge shaped vortex generators are the most efficient. They also concluded that the use of vortex generators has the same effect on the delay of separation as a reduction of the adverse pressure gradient.

The effects of the rotational direction of the vortices have been investigated during the experiments of [Pauley and Eaton, 1988]. The experiments showed that the boundary layer was thickened in the upwash region and thinned in the downwash region. The co-rotating vortex pairs decayed significantly faster than the counter-rotating vortices.

To see the influence of the vortex generators on the efficiency of wind turbines [Gyatt, 1986] looked at the positioning for conventional vortex generators on the turbine blades. A significant increase in the power output was measured when vortex generators were used. The largest gain in power output was found when the vortex generators were used on the full span of the blade.

Besides experimental research into the effects of conventional vortex generators, these effects have also been studied numerically. Different RANS turbulence models were used to simulate the influence of vortices in a turbulent boundary layer by [Liandrat et al., 1987]. In the case of a single vortex the models based on the Boussinesq hypothesis provided a good prediction of the mean flow features, but these simple models failed for a vortex pair. The use of Reynolds Stress Transport Models improved the predictions for the vortex pair, although they still underestimated the Reynolds shear stresses. [Kim and Patel, 1994] used $k-\varepsilon$-models to predict the flow of a common flow down vortex pair. They showed that the mean flow features could be predicted quite well outside the vortex core. For the same configuration [Guohua and Guanghua, 1998] concluded that Reynolds Stress Transport models predict the mean flow field well, but some more complicated features of the Reynolds stress distribution in the vortex cores were not correctly captured.

The common flow up vortex pair has also been investigated with a Large Eddy Simulation by [Liu et al., 1996]. This more accurate computation technique gives more insight in the turbulent structures in the disturbed boundary layer. In the upwash region high levels of turbulent kinetic energy and anisotropy of the normal stresses were found. In their paper the modelling deficiencies of RANS turbulence models are also addressed. The researchers state that due to the high level of anisotropy in the Reynolds stresses a scalar eddy viscosity cannot be accurate and they are not sure if Reynolds Stress Transport Models can correctly capture all effects present in the flow.

For a single micro vortex generator numerical studies have been performed by [Allan et al., 2002] and [Wik and Shaw, 2004]. Both studies showed that the $k-\omega$-SST model predicts the mean flow field quite well, but the Reynolds stress distributions were much better predicted by a Reynolds Stress Transport model. A vortex pair was investigated by [Ashill et al., 2001] and it was shown that the flow was predicted well by CFD up to $10H$ behind the vortex generators.

Experimentally micro vortex generators have been investigated by [Ashill et al., 2001] and more recent by [Baldacchino et al., 2015]. This last research will be discussed in more detail in the next chapter.
Chapter 3

Test case setup

The experiment on a micro vortex generator array by [Baldacchino et al., 2015] provides a detailed insight into the mean flow velocity as well as into the individual components of the Reynolds stress tensor within the turbulent boundary layer. These detailed results will be used to assess the performance of RANS turbulence models. The numerical setup of the RANS simulations is also described in this chapter.

3.1 Experimental setup

Baldacchino investigated the mean flow field and the Reynolds stresses caused by a micro vortex generator with a common flow down configuration by means of Particle Image Velocimetry (PIV). The experiment was conducted in the Boundary Layer Wind Tunnel of the TU Delft, which is capable of producing a relatively thick boundary layer. The wind tunnel has an adjustable back wall which allows for experiments with different pressure gradients. The experiment has been conducted on a zero pressure gradient flow.

A sketch of the layout of the vortex generator pair is shown in Figure 3.1. The freestream velocity $U_\infty$ is equal to 15 m/s and is directed in the direction of the positive $x$-axis. The vane type vortex generators create a common flow down vortex pair and they are arranged in an array. A detailed overview of the setup of this micro vortex generator array is shown in Figure 3.2. The dimensions in that figure are specified in Table 3.1. More detailed information about the experiment can be found in [Baldacchino et al., 2015].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Size in $H$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$H$</td>
<td>5 mm</td>
<td>1</td>
</tr>
<tr>
<td>$H/\delta$</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>$d$</td>
<td>12.5 mm</td>
<td>2.5</td>
</tr>
<tr>
<td>$l$</td>
<td>12.5 mm</td>
<td>2.5</td>
</tr>
<tr>
<td>$D$</td>
<td>30 mm</td>
<td>6</td>
</tr>
<tr>
<td>$\phi$</td>
<td>18°</td>
<td>-</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>15 m/s</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 3.1 List of experimental parameters. [Baldacchino et al., 2015]
3.2 Numerical setup

The numerical simulation of the vortex generators will be done with the open source Computational Fluid Dynamics package OpenFOAM. The steady state solver simpleFoam is used. With this solver the incompressible Reynolds Averaged Navier-Stokes equations (equation 2.4) are solved by means of the SIMPLE algorithm. The spatial derivatives are computed using central differences, which are second order accurate. The convective terms in the RANS equations are discretised by an upwind scheme. The computation was considered converged if the initial residual for \( p \) was below \( 5 \cdot 10^{-4} \) and below \( 1 \cdot 10^{-4} \) for all other quantities.

The setup for the vortex generator flow is a flat plate with a vortex generator that creates a counter-rotating vortex pair with a common flow down configuration. The dimensions of the computational domain are shown in Figure 3.3. The distances are expressed in the height of the vortex generator \( H = 5 \text{ mm} \). Dashed lines indicate lines along which flow variables will be plotted and compared with the experiments from Baldacchino.

The vortex generators are not modelled, but are physically present in the computational mesh as small plates with a no slip boundary condition. The flow area is spatially divided into cells with a structured mesh with 1,465,056 finite volume cells. A close up of the mesh around the vortex generator is shown in Figure 3.4, where a top view and a side view of the vortex generators are presented. Around the vortex generators the mesh is refined to capture the effects of the wall correctly.
Figure 3.3 Dimensions of the vortex generator case

- Top view of the vortex generator pair
- Side view of the vortex generator pair

Figure 3.4 Close up of the computational grid around the vortex generator pair

- (a) Top view of the vortex generator pair
- (b) Side view of the vortex generator pair
The boundary conditions are a no slip condition for the bottom plate and the vortex generators. The inflow patch is an inflow boundary condition where a boundary layer profile is defined with a free stream velocity of $U_\infty = 15.4 \text{ m/s}$ and a boundary layer height $\delta = 3H$. The outflow patch and the top of the domain have a zero gradient boundary condition for the pressure and velocity components. The side planes are symmetry boundaries to make the vortex generator pair part of a larger array of vortex generators.

At the walls standard wall functions are used. In the $k$-$\omega$-SST case $y^+ < 6$ for the bottom plate, the other models have a value of 8 for the bottom plate. This low value for $y^+$ makes sure that almost the whole turbulent boundary layer is simulated, which is important because the vortices occur within the boundary layer.

To speed up the expensive three dimensional computation the initial values in the domain come from a simulation of a turbulent boundary layer which is projected onto the domain.

### 3.3 Turbulence models

In a RANS flow solver the six components of the Reynolds stress tensor $\langle u_i u_j \rangle$ express the influence of the turbulent fluctuations on the mean flow. These unknown components need to be expressed in known flow quantities in a process called turbulence modelling. A broad range of RANS turbulence models have been developed over the years, each with their own degree of complexity and physical realism. For this research four different turbulence models are used. These models are presented in Table 3.2. The turbulence models are categorised in two different groups, based on the type of model. The first three models in Table 3.2 were already available in OpenFOAM. The Speziale-Sarkar-Gatski model (SSG) was not yet available and has been implemented for this case. The models will be shortly discussed in the next paragraphs. A more elaborated overview of the models is given in Appendix A.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Category</th>
<th>In OpenFOAM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard $k$-$\varepsilon$</td>
<td>Eddy viscosity</td>
<td>Yes</td>
</tr>
<tr>
<td>$k$-$\omega$-SST</td>
<td>Eddy viscosity</td>
<td>Yes</td>
</tr>
<tr>
<td>Launder Gibson RSTM</td>
<td>Reynolds stress</td>
<td>Yes</td>
</tr>
<tr>
<td>SSG RSTM</td>
<td>Reynolds stress</td>
<td>No</td>
</tr>
</tbody>
</table>

**Table 3.2** List of the used turbulence models

#### 3.3.1 Standard $k$-$\varepsilon$

A commonly used turbulence model is the standard $k$-$\varepsilon$ model. The most important assumption made in this model is that the Reynolds stresses are aligned with the mean rate of strain, the so called Boussinesq hypothesis. The Reynolds stresses are constructed with the aid a turbulent viscosity $\nu_t$, which acts as extra diffusion in the RANS equation. This turbulent viscosity is a scalar quantity and is computed as

$$
\nu_t = C_\mu \frac{k^2}{\varepsilon},
$$

where $C_\mu$ is a model constant with value 0.09, $k$ the turbulent kinetic energy and $\varepsilon$ the scalar dissipation rate. For the last two quantities an additional modelled transport equation is
solved each time step. The model equations and constants can be found in the original paper by [Launder and Spalding, 1974]. The standard \( k-\varepsilon \) model is widely used in the industry, because of its robustness, although it is known to fail in a number of cases.

3.3.2 \( k-\omega \)-SST

The \( k-\omega \)-SST model is also based on the Boussinesq hypothesis, but uses a different turbulent quantity, \( \omega \), instead of \( \varepsilon \), because the transport equation of the latter is very hard to integrate all the way to a solid boundary.

The \( k-\omega \)-SST model is a zonal model where close to a wall the original \( k-\omega \) model from Wilcox is used. Further from the solid boundaries the model blends into the standard \( k-\varepsilon \) model. The switch between the two models is done with blending functions based on the distance from the wall. The SST model also puts a limit on the shear stress in the boundary layer. The assumptions and the model equations are further explained in the paper of [Menter, 1993].

3.3.3 Launder Gibson RSTM

In order to include mode physics into turbulence models, the so called Reynolds Stress Transport Models (RSTM) have been developed. These models do not relate the Reynolds stresses to the local velocity gradient, but solve a transport equation for each of the six components of the Reynolds stress tensor. This increases the computation time considerably, but more physical processes can be modelled. The redistribution term in the modelled transport equation contributes to the anisotropy of the Reynolds stress tensor and is therefore one of the key features of the RST models. The Launder Gibson RSTM uses the distance to the wall to include the reflection of Reynolds stresses by the wall. [Gibson and Launder, 1978]

3.3.4 Speziale Sarkar Gatski RSTM

The Speziale Sarkar Gatski RSTM uses a non linear model for the redistribution term. [Speziale et al., 1991] This non linear terms makes the use of the wall distance no longer necessary, which makes this model more applicable for complex geometries.
Chapter 4

Results

The results from the numerical simulations are discussed and compared to the experimental data. The differences between the used turbulence models will be discussed. The results for the two Reynolds stress models are so close together that only the results for the SSG model will be presented.

In Table 4.1 the number of iterations the SIMPLE algorithm used to reach a converged solution for each turbulence model is presented. Within each SIMPLE-iteration each flow and turbulent quantity is solved separately. Each quantity is also solved by means of an iterative process. This results in a number sub iterations that are not shown in the table.

Compared to the Reynolds Stress Transport Model the eddy viscosity models converge with less iterations, just as expected. The Reynolds stress models are less robust, because they need an initialisation with 100 $k$-$\epsilon$ iterations to come to a stable solution. These 100 iterations are already included in the total number of iterations.

To compare the computational cost of the models, their total runtime is compared to the total runtime of the $k$-$\omega$-SST model. The total runtime for each model is computed by considering the average time of one SIMPLE iteration based on 100 iterations with the model. This relative runtime shows that the cost of the standard $k$-$\epsilon$ model comes close to that of the Reynolds Stress Transport Models.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Iterations</th>
<th>Initialisation</th>
<th>Relative runtime</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard $k$-$\epsilon$</td>
<td>533</td>
<td>-</td>
<td>1.25</td>
</tr>
<tr>
<td>$k$-$\omega$-SST</td>
<td>494</td>
<td>-</td>
<td>1</td>
</tr>
<tr>
<td>Launder Gibson RSTM</td>
<td>586</td>
<td>100 $k$-$\epsilon$ iterations</td>
<td>1.30</td>
</tr>
<tr>
<td>SSG RSTM</td>
<td>590</td>
<td>100 $k$-$\epsilon$ iterations</td>
<td>1.42</td>
</tr>
</tbody>
</table>

Table 4.1 Number of iterations and relative runtime for the different turbulence models

4.1 Mean flow

For the assessment of the mean flow the streamwise velocity component, $U_x$, is plotted in Figure 4.1 over the height of the boundary layer at different $y$-locations. These positions
are shown in Figure 3.3. The positions are in the middle between the vortex generator pair \( y = 0 \), at the trailing edge of the vortex generator \( y = 1.25H \), \( y = 2H \) and at the symmetry plane \( y = 3H \).

![Figure 3.3](image)

**Figure 3.3** Positions of vortex generator pair (middle between vortex generator, at trailing edge, at symmetry plane).

(a) \( U_x/U_\infty \) at \( x = 10H \)

(b) \( U_x/U_\infty \) at \( x = 50H \)

**Figure 4.1** Profiles of normalised streamwise velocity \( U_x/U_\infty \) at \( x = 10H \) and \( x = 50H \) at positions \( y = [3, 2, 1.25, 0, -1.25, 2, 3]H \)

From the velocity profiles it is clear that the profiles become fuller close to the wall compared to the undisturbed turbulent boundary layer. Only at \( y = \pm 3H \) the \( U_x \) profile is less full. The simulations with the RANS models show a larger deficit at \( y = \pm 3H \) than the experimental data. This is especially true further downstream. The Reynolds stress models seem to predict the profiles the best.

The positive effect of the micro vortex generators are more local than for conventional vortex generators, so the decay of the vortices needs to be predicted more accurate. To see the decay of the vortices the streamwise vorticity \( \omega_x \) is considered next. Figure 4.2 shows contours of the streamwise vorticity \( \omega_x \) at three different planes downstream of the vortex generator. From the experimental data it is clear that the vortices become weaker downstream and move apart.

The contours clearly show that the standard \( k-\varepsilon \)-model does not predict the correct shape of the vortices, although the decay of the vortices is quite well predicted by the model. The RSTM and the \( k-\omega \)-SST model do not differ much when the vorticity contours are compared. Both models predict clear regions of vorticity at \( x = 50H \), however this is less pronounced in the experimental data.
Figure 4.2  Contour plots for the streamwise vorticity $\omega_x$ at $x = [10H, 25H, 50H]$. 
4.2 Turbulent quantities

Of the turbulent quantities the turbulent kinetic energy $k$ is the most widely used to address the intensity of the turbulent fluctuations. Figure 4.3 shows the value of $k$ at different planes downstream. The experimental data suggests that close to the trailing edge of the vortex generators the turbulent kinetic energy peaks at the vortex cores. Further downstream the peak in $k$ shifts to the region above the vortex cores.

None of the turbulence models is able to accurately reproduce the turbulent kinetic energy distribution at $x = 10H$. The peak of $k$ in the vortex cores is not present. The RSTM shows a better match for $k$ further downstream.
Figure 4.3  Turbulent kinetic energy $k$ at $x = [10H, 25H, 50H]$. Contours of the streamwise vorticity $\omega_x$ with contours $[-500, -400, -300, -200, -100, 100, 200, 300, 400, 500]$ are also shown.
The turbulent kinetic energy only gives information about the Reynolds normal stresses. To see what is happening with the Reynolds shear stresses the magnitude of the shear stresses is defined as

\[ \sigma = \sqrt{\langle uv \rangle^2 + \langle uw \rangle^2 + \langle vw \rangle^2} \]

\[ = \sqrt{R^2_{xy} + R^2_{xz} + R^2_{yz}}. \] (4.1)

This measure, shown in Figure 4.4, shows regions in the flow where shear stresses are present. The shear stresses are mainly of importance above the vortex core. The results from the numerical simulations are also presented in the figure. The RSTM shows the best resemblance with the experimental data. The \( k-\omega \)-SST model underpredicts the intensity of the Reynolds shear stresses.
Figure 4.4 Magnitude of the Reynolds shear stresses at \( x = [10H, 25H, 50H] \). Contours of the streamwise vorticity \( \omega_x \) with contours \([-500, -400, -300, -200, -100, 100, 200, 300, 400, 500]\) are also shown.
A very interesting observation can be made when the Reynolds stress anisotropy is addressed. The magnitude of anisotropy is assessed by considering the invariants of the Reynolds stress anisotropy tensor $b_{ij}$ as defined in paragraph 2.1. The second and third invariant at different downstream positions of the vortex generator are shown in Figure 4.5 and Figure 4.6 respectively. When the experimental data is compared to the undisturbed boundary layer as shown in Figure 2.3, it becomes clear that the vortices influence the Reynolds stress anisotropy in the boundary layer. In the downwash region and the vortex cores the turbulence becomes more isotropic, $\Pi_b$ and $\Pi_{III_b}$ are close to zero, in contrast to the region just above the vortices where the anisotropy is enlarged.

The turbulence models that use an eddy viscosity show anisotropy in the correct regions of the flow, but the degree of anisotropy is too low. This is a known deficiency of RANS turbulence models based on the Boussinesq hypothesis. The Reynolds stress model does not have this limitation and predicts a much higher degree of anisotropy in the flow, although the anisotropy is higher than in the experiments.
Figure 4.5 Second invariant $II_b$ of the Reynolds stress anisotropy tensor $b_{ij}$ at $x = [10H, 25H, 50H]$. Contours of the streamwise vorticity $\omega_x$ with contours $[-500, -400, -300, -200, -100, 100, 200, 300, 400, 500]$ are also shown.
Figure 4.6 Third invariant $III_b$ of the Reynolds stress anisotropy tensor $b_{ij}$ at $x = [10H, 25H, 50H]$. Contours of the streamwise vorticity $\omega_x$ with contours $[-500, -400, -300, -200, -100, 100, 200, 300, 400, 500]$ are also shown.
Chapter 5

Conclusion and recommendations

Vortex generators affect the flow over a curved surface by creating streamwise vortices. These vortices make the streamwise velocity profile fuller close to the wall. This enables the flow to overcome larger adverse pressure gradients without the flow separating from the surface. The different designs of vortex generators determine the efficiency of the devices to enhance the velocity profile. A more recent trend in the design of vortex generators is the development of micro vortex generators in order to reduce the additional drag created by the generators. Micro vortex generators have a more local positive influence on the flow. This increases the need for accurate modelling of the effects of the generators. In this thesis the effects of micro vortex generators on the flow were modelled using different RANS turbulence models and the results were compared to experimental data.

5.1 Conclusion

For the mean flow the predictions made by the $k$-$\omega$-SST model and the RSTM were in line with the experimental data. The standard $k$-$\varepsilon$ model was less successful in predicting the vortex shape. This confirms the results found for conventional vortex generators. The decay of the vortex is predicted quite well by all models, which is important because the positive influence of the vortices on the streamwise velocity profile is more local for the micro vortex generators.

When the turbulent kinetic energy was considered the RANS models failed to reproduce the intensity of this quantity close to the vortex cores. Further downstream the predictions of the RSTM were in better correspondence with the experimental data. The magnitude of the Reynolds shear stresses was also investigated. This showed that again the predictions of the RSTM are the closest to the experimental data. The $k$-$\omega$-SST model underpredicted the magnitude of the Reynolds shear stresses.

The Reynolds stress anisotropy found in the experiment showed that the vortex suppresses the turbulent anisotropy in the vortex core and the downwash region. This effect is partly captured by the turbulence models, but the degree of anisotropy is either underestimated (eddy viscosity models) or overestimated (RSTM).

From this the conclusion can be drawn that the standard $k$-$\varepsilon$ model is not the best choice for the study of the flow of micro vortex generators as it does not predict the correct shape of the
vortices. When the mean flow features are of interest the best choice is the $k$-$\omega$-SST model, because the computation time of this model is much less than the RSTM, but the mean flow is predicted with almost the same accuracy. The RSTM predicts the Reynolds stresses the most precise, but there are still some differences with the experimental results that could be improved.

5.2 Recommendations

A disadvantage of the RANS approach is the need for turbulence models. Although it has been shown that the more advanced models can predict the flow quantities quite well, there are still some deficiencies. A more physical approach would be to perform a Large Eddy Simulation to confirm the behaviour that is observed from the experiments. The advantage of a LES is that it can assess information about terms that are modelled in RANS turbulence models. Information about these modelled terms can be used to improve the RANS model for vortex generators.

As the RSTM predicts a too high degree of anisotropy and the eddy viscosity models under-predict the anisotropy, a further approach could be to construct a hybrid model that includes information from the Reynolds Stress Transport Model in a simpler model. This would also reduce the computational cost.

The computation time for the numerical simulation of the flow over a micro vortex generator could be also reduced by considering a RSTM within the boundary layer and for example the standard $k$-$\varepsilon$ model in the freestream. This is possible because the influence of the vortices is confined to the boundary layer. The reduction of computational time is possible because the boundary layer comprises only a small part of the computational domain. Outside the boundary layer the turbulent quantities are zero so a Reynolds Stress Transport Model is not necessary.
Bibliography


Appendix A

Turbulence models

For the assessment of the flow field over a pair of micro vortex generators several different RANS turbulence models have been used. These models differ in their complexity, computational cost and physical realism. The used models and their equations are shortly discussed in this chapter.

A.1 Standard $k$-$\varepsilon$

The standard $k$-$\varepsilon$ model is widely used turbulence model and was developed in the 1970s by [Launder and Spalding, 1974]. For this eddy viscosity model the underlying assumption is the Boussinesq hypothesis, which relates the Reynolds stresses to the mean rate of strain by means of an eddy viscosity

$$\langle u_i u_j \rangle = \frac{2}{3} k \delta_{ij} + 2 \nu_t S_{ij}.$$  \hspace{1cm} (A.1)

The eddy viscosity is computed as

$$\nu_t = C_\mu \frac{k^2}{\varepsilon},$$ \hspace{1cm} (A.2)

where $C_\mu$ is a model constant with value 0.09, $k$ the turbulent kinetic energy and $\varepsilon$ the scalar dissipation rate. The modelled transport equation for the turbulent kinetic energy is given by

$$\frac{Dk}{Dt} = P_k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial k}{\partial x_j} \right] - \varepsilon.$$ \hspace{1cm} (A.3)

In which the production term $P_k = -\langle u_i u_j \rangle \frac{\partial u_i}{\partial x_j}$ and $\sigma_k$ is a model constant. The dissipation rate $\varepsilon$ of turbulent kinetic energy is determined from its own modelled transport equation

$$\frac{D\varepsilon}{Dt} = C_{\varepsilon 1} \frac{\varepsilon}{K} P_k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] - C_{\varepsilon 2} \frac{\varepsilon^2}{K}.$$ \hspace{1cm} (A.4)

The values for the model constants can be found in Table A.1.

A.2 $k$-$\omega$-SST

The $k$-$\omega$-SST model developed by [Menter, 1993] is also based on the Boussinesq hypothesis, but uses a different turbulent quantity, $\omega$, instead of $\varepsilon$. The transport equation of the latter is
very hard to integrate all the way to a solid boundary. The transport equation for \( \omega \) is given by

\[
\frac{D\omega}{Dt} = \frac{\gamma}{\nu_t} P_k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] - \beta \omega^2 + 2 \left( 1 - F_1 \right) \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}.
\]  

(A.5)

In this equation the model parameters are blended between two values by means of the \( F_1 \) blending function. This function makes sure that in the boundary layer the parameters from the original \( k-\omega \) model of Wilcox is used. Outside the boundary layer the equation is effectively equal to the \( \varepsilon \) equation but then rewritten in terms of \( \omega \).

The use of \( \omega \) also alters the modelled equation for the turbulent kinetic energy to

\[
\frac{Dk}{Dt} = P_k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_R} \right) \frac{\partial k}{\partial x_j} \right] - \beta^* \omega k.
\]  

(A.6)

For the \( k-\omega \)-SST model the turbulent viscosity is retrieved from

\[
\nu_t = \frac{a_1 k}{\max(a_1 \omega; SF_2)}.
\]  

(A.7)

Where \( a_1 \) is a model constant and \( S = \sqrt{S_{ij} S_{ij}} \). This formulation restricts the value for the shear stresses in the boundary layer to be proportional to \( k \). The blending function \( F_2 \) makes sure that outside the boundary layer unrestricted values for \( \omega \) are used. This restriction of the shear stresses and the use of \( \omega \) instead of \( \varepsilon \) near the wall, improves the prediction of flows with separation.

### A.3 Launder Gibson RSTM

In order to include mode physics into turbulence models, the so called Reynolds Stress Transport Models (RSTM) have been developed. Instead of using the Boussinesq hypothesis, a modelled transport equation for the Reynolds stress tensor is solved. This modelled equation is given by

\[
\frac{D \langle u_i u_j \rangle}{Dt} = P_{ij} + \frac{\partial}{\partial x_i} \left[ \left( \nu + \frac{\nu_t}{\sigma_R} \right) \frac{\partial \langle u_i u_j \rangle}{\partial x_i} \right] - \frac{2}{3} \varepsilon \delta_{ij}.
\]  

(A.8)

Where each of the terms on the right hand side represents a different physical process: production of Reynolds stresses, the redistribution of energy between the individual components, diffusion and dissipation, respectively. Model developers have focussed on the redistribution term, due to the fact that the degree of anisotropy of the Reynolds stresses is largely determined by this term. [Al-Sharif, 2011]

In case of the Launder Gibson RSTM the redistribution term is split according to

\[
\phi_{ij} = \phi_{ij}^s + \phi_{ij}^r + \phi_{ij}^{w,s} + \phi_{ij}^{w,r}.
\]  

(A.9)
The first two terms of the above equation are the slow and rapid pressure terms. The latter two terms are the contributions from walls in the domain on the slow and rapid terms. These terms include information about the wall normal direction \(n_i\) and the distance to the wall to include the reflection of Reynolds stresses by the wall. [Gibson and Launder, 1978] All four contributions to the redistribution term in this model are given by

\[
\begin{align*}
\phi_{ij}^s &= -2C_1 \varepsilon b_{ij}, \\
\phi_{ij}^r &= -C_2 \left( P_{ij} - \frac{2}{3} P_k \delta_{ij} \right), \\
\phi_{ij}^{w,s} &= C_w^1 \varepsilon \left( (u_k u_m) n_k n_m \delta_{ij} - \frac{3}{2} (u_i u_k) n_k n_j - \frac{3}{2} (u_j u_k) n_k n_i \right) f_w, \\
\phi_{ij}^{w,r} &= C_w^2 \left( \phi_{km} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{ik} n_k n_j - \frac{3}{2} \phi_{jk} n_k n_i \right) f_w.
\end{align*}
\]

These terms are linear combinations of the Reynolds stress tensor, the mean velocity gradients, the scalar dissipation rate and the wall normal direction \(n_i\). The function \(f_w\) depends on the wall distance.

### A.4 Speziale Sarkar Gatski RSTM

The Speziale Sarkar Gatski RSTM uses a model for the redistribution term that is non linear in the Reynolds stresses. [Speziale et al., 1991] This non linear terms makes the use of the wall distance no longer necessary, which makes this model more applicable for complex geometries. The total redistribution term of this model is given by

\[
\phi_{ij} = - (C_1 \varepsilon + C_1^* P_k) b_{ij} + C_2 \varepsilon \left( b_{ik} b_{kj} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right) + (C_3 - C_3^* \sqrt{\Pi_6}) k S_{ij} + C_4 k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right) + C_5 k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right),
\]

where

\[
\begin{align*}
S_{ij} &= \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right), \\
\Omega_{ij} &= \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right).
\end{align*}
\]