Numerical study of turbulent flow over an S-shaped hydrofoil
T Micha Prem Kumar and Dhiman Chatterjee
DOI: 10.1243/09544062JMES929
The online version of this article can be found at:
http://pic.sagepub.com/content/222/9/1717

Published by:
SAGE
http://www.sagepublications.com

On behalf of:
Institution of Mechanical Engineers

Additional services and information for Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science can be found at:

Email Alerts: http://pic.sagepub.com/cgi/alerts
Subscriptions: http://pic.sagepub.com/subscriptions
Reprints: http://www.sagepub.com/journalsReprints.nav
Permissions: http://www.sagepub.com/journalsPermissions.nav
Citations: http://pic.sagepub.com/content/222/9/1717.refs.html

>> Version of Record - Sep 1, 2008
What is This?
Numerical study of turbulent flow over an S-shaped hydrofoil

T Micha Prem Kumar and Dhiman Chatterjee

Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai 600 036, India

The manuscript was received on 2 November 2007 and was accepted after revision for publication on 17 April 2008.

DOI: 10.1243/09544062JMES929

Abstract: In this paper, a numerical study of turbulent flow over the S-shaped hydrofoil at 0° angle of attack has been reported. Here, the flow takes place over concave and convex surfaces and is accompanied by the favourable and adverse pressure gradients and flow separation. Modelling such a flow poses a formidable challenge. In the present work four turbulence models, namely, $k-\varepsilon$ realizable, $k-\omega$ shear stress transport, $\nu^2-f$, and the Reynolds stress model (RSM) were examined. Simulations were performed on a structured grid using finite-volume method formulation. Commercial software was used for grid generation and numerical simulation. A comparison of the experimental data of Madhusudan et al. (Fluid Dyn. Res., 1994, 14(5), 241–258) and numerical predictions were made, and the suitability of turbulence models was ascertained for both the mean and turbulent quantities. It was seen that $\nu^2-f$ works better for predicting the mean quantities and the RSM for turbulent quantities.

Keywords: turbulent boundary layer, turbulence model, hydrofoil, pressure gradient effect, curvature effect, numerical simulation

1 INTRODUCTION

The turbulent flows that advance over the surface with curvature normal to the mean flow direction are common in many engineering applications. The effects of streamline curvatures on turbulent flows in many situations can be quite large [1].

In the past, the experimental studies of both internal flows within curved ducts and external flow around curved surfaces have been carried out [2–5]. From these careful experiments, it is evident that the turbulent shear stress is damped on convex surfaces [2], while the turbulent intensities inside the boundary layer increase in the presence of concave surfaces [3, 4]. Baskaran et al. [5, 6] studied the effect of the curvature and pressure gradient on the behaviour of turbulent flows. They showed that the shear stress is more sensitive to streamline curvature than the turbulent kinetic energy.

In the past, some efforts were made in solving the boundary-layer equations using different turbulent models. Gibson et al. [7] calculated the turbulent boundary layers on the curved surfaces by solving the boundary-layer equation with measured longitudinal pressure gradient, together with a Reynolds stress turbulence model and wall functions. In reality, the surface curvature effects are often accompanied by the pressure gradient effects and these need to be accounted for.

Richmond and Patel [8] carried out a numerical simulation of the flow inside a curved duct as well as an external flow over convex and concave surfaces. They used the $k-\varepsilon$ model. Pressure distribution and velocity profile predictions matched well but the skin-friction coefficient and Reynolds stress data differed considerably from that of the experimental data.

Luo and Lakshminarayana [9, 10] investigated the effects of strong convex and concave curvatures on the turbulent duct flow. They found that the Reynolds stress model (RSM) could successfully capture the damping of turbulence near the convex wall but underpredicted enhancement of turbulence near the concave wall. The difference in the relative success of the model to capture convex and concave curvatures may be attributed to the attenuation of existing eddies.
by the convex structure and reorganization of eddy structures in the concave curvature.

Davidson [11] had predicted the flow around an aerofoil using the RSM. He concluded that in comparison with the two-layer $k-\varepsilon$ model, the RSM could predict stall and flow features better. Rumsey and Gatski [12] studied the capabilities of three different turbulent models for predicting the effect of curvature on flow. They concluded that none of the one- or two-equation linear eddy viscosity models (EVM) and the algebraic stress model they had employed could fully capture the suppressed turbulence near the convex wall or the enhanced turbulence near the concave wall. The RSM could capture the suppression of turbulence. Sleiti and Kapat [13] compared the performance of two-equation eddy viscosity and RSM for simulating flow and heat transfer in rotating rib-roughened internal cooling channels. They concluded that overall the RSM performed better than the EVM. Jing-lei et al. [14] used linear and non-linear EVM, a quadratic algebraic stress model, and the RSM for predicting flows having strong curvature effects. Yakinthos et al. [15] simulated flow in a $90^\circ$ rectangular duct by applying three low-Reynolds-number turbulence models—two EVMs and a RSM. In general, the RSM was seen to give results consistent with the experimental values, but its predictive capability was poor near the concave wall. From the above studies, it is clear that the effects of curvature on the velocity profiles and turbulence intensity must be predicted accurately to capture the boundary layer development over the curved surface.

In contrast to the conventional symmetric or single cambered aerofoil, attention is given here to the flow over a double-cambered S-shaped hydrofoil. This type of aerofoil profile is used in runner blades of the fully reversible pump-turbine that finds application in tidal structures in the concave curvature.

These blades are represented by a four-digit number preceded by $S$ [16]. The first and second digits represent the maximum camber and maximum thickness, respectively, as percentage of chord, and the last two digits together give the location of maximum camber as percentage of chord from either end. In the present work, the S3525 profile, which may be better suited as a runner blade in a reversible pump-turbine [17], has been used.

An accurate prediction of the flow past an S-blade will depend on the choice of the turbulence model. It is a known fact that there is no single turbulence model that can be applied for all flow situations. So, it becomes necessary to examine the predictive capabilities of some of the possible turbulence models against the experimental data available.

Strictly speaking, the only fundamentally secure approach to simulate turbulent flow is to use direct numerical simulation (DNS) that can resolve the entire turbulent spectrum. Alternatively, large eddy simulations (LES) may be employed, in which the smaller eddies are filtered and are modelled using a subgrid scale model, whereas the larger energy carrying eddies are simulated. Naturally, DNS requires larger computational resources than the LES. However, for most practical problems, not only the DNS but also the LES poses a prohibitively large computational resource because near a wall, large dynamically influential turbulent scales are small, and so a very dense mesh is required [18]. For most practical problems, therefore, Reynolds-averaged Navier–Stokes (RANS) equations are employed with appropriate closure models to simulate the flow.

There are two broad classifications of RANS, namely, EVM and RSM. Two of the most well-known and popular RANS models are $k-\varepsilon$ and $k-\omega$, and their variants [19]. Durbin [20] successfully used the $k-\varepsilon-v^2$ model for the turbulent separated flows over a backward facing step, in a plane diffuser and also around a triangular cylinder. Good agreement with the experimental result was found, in this case, of massive separation and smooth separation. Iaccarino [21] carried out the computation of turbulent flow in an asymmetric two-dimensional diffuser and reported that the result obtained using the $v^2-f$ model was consistent with the experimental data.

In this study, a numerical simulation of a two-dimensional turbulent flow over an S-blade for four different turbulence models has been carried out. These models are $k-\varepsilon$ realizable, $v^2-f$, $k-\omega$ shear stress transport (SST), and the RSM. From the survey of literature on turbulence modelling available, it is clear that the EVMs find application in a wide range of practical problems and the RSM is particularly useful in predicting flows, where anisotropy could be significant. This has also led to the inclusion of these models.
in commercial computational fluid dynamics (CFD) software.

The main objective of the work is to compare the predictive capabilities of these models for the simulation of turbulent flow over surfaces having consecutive concave and convex curvatures. The numerical methods and turbulence models with the appropriate equations are presented in section 2. Simulation procedure including grid generation is also covered in section 2, results and discussions are given in section 3, and conclusion in section 4.

2 GOVERNING EQUATIONS AND NUMERICAL SCHEME

The governing equations for steady, incompressible fluid are

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

\[
\frac{\partial u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (v + u_i) \frac{\partial u_i}{\partial x_i} - \frac{1}{\rho} \frac{\partial P}{\partial x_i} \right]
\]  

Instantaneous Navier–Stokes equations can be time averaged, in order to transform it into the RANS equation in such a way that the small-scale turbulent fluctuations do not have to be simulated directly. This transformation introduces an additional Reynolds stresses \(-\rho u_i u_j\) term in the governing equation that needs to be modelled in order to achieve 'closure'. This is achieved through the different turbulence models. The next section is an introduction about the different turbulent models used in this work.

2.1 Turbulence modelling

Different turbulent models are available and, based on the existing knowledge of 'success' and 'failure' of these models in solving the different flow situations, four models have been identified for the present work, which involves the flow where streamline curvature, pressure gradient effects can be significant and where flow separation is expected. In the order of increasing complexity, these are \(k-\varepsilon\) realizable, \(k-\omega\) SST, \(k^2-f\), and the RSM. These models are briefly described in the next subsection. The first three models are the EVM, whereas the last one is a second moment closure model.

2.1.1 \(k-\varepsilon\) realizable model

The modelled transport equations for \(k\) and \(\varepsilon\) in the \(k-\varepsilon\) realizable model are [22]

\[
\frac{\partial}{\partial x_j} (ku_j) = \frac{\partial}{\partial x_j} \left[ (v + u_j) \frac{\partial k}{\partial x_j} \right] + P - \varepsilon
\]
A blending function $\lambda_\epsilon$ is defined in such a way that it is equal to unity far from the walls and is zero very near to the walls. The blending function chosen is

$$\lambda_\epsilon = \frac{1}{2} \left[ 1 + \tanh \left( \frac{\text{Re}_y - \text{Re}_y^+}{A} \right) \right]$$ (12)

The constant $A$ determines the width of the blending function. It is defined such that the value of $\lambda_\epsilon$ will be within 1 per cent of its far-field value, given a variation of $\Delta \text{Re}_y$.

The $\epsilon$ field is computed from

$$\epsilon = \frac{k^{3/2}}{l_i}$$ (13)

The length scale $l_i = yC_i (1 - e^{-\text{Re}_y/A})$.

The constants in the length scale formula are

$$C_i = kC_{\mu}^{-3/4}, \quad A_\mu = 70, \quad A_e = 2C_i$$

### 2.1.2 $\nu^2-f$ model

The $\nu^2-f$ turbulence model is an alternative to the $k-\epsilon$ model and was introduced to model the near-wall turbulence without the use of exponential damping or wall functions [21]. The turbulence model described earlier uses turbulence kinetic energy, $k$, as the velocity scale in the eddy viscosity region. However, it is well known that the turbulence is anisotropic near the wall, and so $k$ is not a good representative of the velocity scale. Stress shear is driven by the normal stress, $\nu^2$, in the near-wall layer. In the enhanced model formulation described earlier, a damping function is used to account for more rapid decay in wall-normal intensity than that of the turbulent energy. This damping is assumed to be purely viscous and in reality, viscous damping occurs over a thinner region than is implied in the damping function [18]. To overcome these difficulties, the $\nu^2-f$ model was developed by Durbin [20]. It incorporates a simplified transport equation for the normal stress perpendicular to the streamlines (or, the wall) that serve as the turbulent-velocity scale in the eddy viscosity, in preference to the turbulence energy. The model requires the solution of four differential equations

$$u_i \frac{\partial k}{\partial x_i} = P - \epsilon + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial k}{\partial x_j} \right)$$ (14)

$$u_i \frac{\partial \epsilon}{\partial x_i} = f_1 C_1 P - f_2 C_2 \epsilon + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \epsilon}{\partial x_j} \right)$$ (15)

The eddy viscosity is obtained from

$$\nu_t = C_{\nu} f_3 \nu T$$ (16)

where

$$f_1 = 1 + 0.045 \left( \frac{k}{\nu^2} \right)^{1/2}$$

$$f_2 = 1$$

$$f_3 = \frac{\nu^2}{k}$$

$$T = \max \left[ \frac{k}{\epsilon}, 6 \left( \frac{\nu}{k} \right)^{1/2} \right]$$

The equations model the turbulence velocity scale $\nu^2$ and its production, $k f$

$$u_i \frac{\partial \nu^2}{\partial x_i} = kf - 6 \nu^2 \epsilon + \frac{\partial}{\partial x_j} \left[ \nu \frac{\sigma_k}{\nu} \frac{\partial \nu^2}{\partial x_j} \right]$$ (21)

$$f = L^2 \frac{\partial^2 f}{\partial x_j \partial x_j} = C_1 \left[ \frac{2/3 - \nu^2/k}{T} \right] + C_2 \frac{P}{k} + \frac{5
}
respectively. $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$, respectively, due to turbulence. $D_\omega$ represents the cross-diffusion term.

For all the EVM discussed above, the turbulent stresses $u'^2$, $v'^2$, and $u'v'$ are calculated based on the Boussinesq assumption.

### 2.1.4 Reynolds stress model (RSM)

Most models used in engineering practice employ the Boussinesq approximation, where the turbulent stresses in the Reynolds-averaged momentum and energy equations are assumed equal to the product of the isotropic eddy viscosity coefficient and mean velocity strain rate. However, when the Boussinesq assumption is not valid, Reynolds stress calculations will be in error regardless of how accurately the model equations are solved. The RSM, which does not use the Boussinesq assumption, is considered to be of a higher level of closure that is available. These models are superior to the algebraic, one- and two-equation models since they eliminate the assumption that the turbulence is locally homogeneous and in equilibrium. These models are currently too CPU-intensive for most engineering calculations, and have yet to be applied to many complex flows, particularly wall-bounded flows. In addition, numerical stability problems may arise due to the absence of turbulent viscosity, making the application in complicated situations difficult at present. In the RSM, the Boussinesq assumption is not used but a partial differential equation (transport equation) for the stress tensor is derived from the Navier–Stokes equation. The exact transport equations for the transport of the Reynolds stresses, $\rho u'_i u'_j$, may be written as follows [22]

\[
\frac{\partial}{\partial x_k} \left( \rho u'_i u'_j \right) = \frac{\partial}{\partial x_k} \left[ \rho u'_i u'_k + p \left( \delta_{ij} u'_i + \delta_{ik} u'_k \right) \right] + \frac{\partial}{\partial x_k} \left( \mu \frac{\partial u'_i}{\partial x_k} \right) + \frac{\partial}{\partial x_k} \left( \mu \frac{\partial u'_j}{\partial x_k} \right) - \rho \left( \frac{\partial u'_i}{\partial x_k} + \frac{\partial u'_k}{\partial x_k} \right) + \rho \left( \frac{\partial u'_j}{\partial x_k} + \frac{\partial u'_k}{\partial x_k} \right) + p \left( \frac{\partial u'_i}{\partial x_k} + \frac{\partial u'_k}{\partial x_k} \right) - 2 \mu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}.
\]

### Modelling turbulent diffusive transport

\[
D_{ij} = \frac{\partial}{\partial x_k} \left( \frac{\mu_i}{\sigma_k} \frac{\partial u'_j}{\partial x_k} \right)
\]

The turbulent viscosity, $\mu_t$, is computed in a way similar to that of $k$-$\epsilon$ model and $\sigma_k = 0.82$.

### Modelling the pressure–strain term

The pressure–strain term contains: $\phi_{ij,1}$, the slow pressure–strain term, also known as the return-to-isotropy term; $\phi_{ij,2}$, the rapid pressure–strain; and $\phi_{ij,w}$, the wall-reflection term.

\[
\phi_{ij} = \phi_{ij,1} + \phi_{ij,2} + \phi_{ij,w}
\]

The slow pressure–strain term, $\phi_{ij,1}$, is modelled as

\[
\phi_{ij,1} = -C_1 \rho \frac{\epsilon}{k} \left( u''_{ij} - \frac{2}{3} \delta_{ij} k \right)
\]

$C_1 = 1.8$

The rapid pressure–strain term, $\phi_{ij,2}$, is modelled as

\[
\phi_{ij,2} = -C_2 \left[ (P_{ij} - C_{ij}) - \frac{2}{3} \delta_{ij} (P - C) \right]
\]

$C_2 = 0.6$

$P_{ij}$ and $C_{ij}$ are defined as in equation (26)

\[
P = \frac{1}{2} P_{kk}, \quad C = \frac{1}{2} C_{kk}
\]

The wall-reflection term, $\phi_{ij,w}$, is responsible for the redistribution of normal stresses near the wall. It tends to damp the normal stress perpendicular to the wall, while enhancing the stresses parallel to the wall. This term is modelled as

\[
\phi_{ij,w} = C_i \rho \left( 1 - \frac{3}{2} \frac{u'_i u'_n n_m \delta_{ij}}{C_i n_l \epsilon d} \right) \left( 3 \frac{u'_i n_k n_l}{C_i n_l \epsilon d} \right)^{3/2} + \frac{2}{3} \phi_{n,k,2,1} n_j n_m 
\]

where $C_i = 0.6$, $C_2 = 0.5$, $n_k$ is the $x_k$ component of the unit normal to the wall, $d$ is the normal distance to the wall, and $C_i = C_i^{3/4} / k$, where $C_{i0} = 0.09$ and $k$ is the von Karman constant ($= 0.4187$).
The dissipation tensor, \( \varepsilon_{ij} \), is modelled as

\[
\varepsilon_{ij} = \frac{2}{3} \delta_{ij} \rho \varepsilon
\]  

(32)

The scalar dissipation rate \( \varepsilon \) is computed with a model transport equation similar to that used in the \( k-\varepsilon \) model.

### 2.2 Numerical scheme

A finite-volume code (Fluent\textsuperscript{®}) solves the discretized equations in a segregated manner, with the SIMPLEC (semi-implicit method for pressure-linked equation-consistent) algorithm. In the SIMPLEC algorithm, the continuity equation (equation (1)) is converted into a discrete Poisson equation for pressure. The differential equations are linearized and solved implicitly in sequence: starting with the pressure equation (predictor stage), followed by the momentum equations and pressure correction equation (corrector stage). The equations for the scalars (turbulent quantities) are solved after updating both the pressure and velocity components. In Table 1, a summary of the numerical parameters used for the computations are reported.

### 2.3 Statement of the problem

A two-dimensional computational domain is used in the present case. The chord length of the hydrofoil is 0.3 m. The inlet boundary condition is kept at a distance 10 times the chord length upstream of the leading edge of the hydrofoil. The height of the domain above and below the profile is six times the chord length. The outlet boundary condition is kept at a distance 10 times the chord length upstream of the profile. Simulation of flow in the domain was carried out subjected to the boundary conditions described below, and both the mean and turbulent quantities were predicted. The numerical predictions were then compared with the experimental data available.

### 2.3.1 Boundary condition

Uniform flow velocity of 22.5 m/s was imposed at the velocity inlet and atmospheric pressure was prescribed at the pressure outlet condition of the domain. No-slip boundary conditions were set on the hydrofoil surface and wall. The values of turbulence intensity and length scale were obtained from the relations \( 0.13 \times Re^{1/8} \) and \( 0.07 \times D_h \), respectively, where \( D_h \) is the hydraulic diameter and \( Re \) is the Reynolds number based on the chord length and velocity at the inlet, and its value is \( 4.5 \times 10^5 \). Air was used as the working medium for simulation. All the simulations were done at 0° angle of attack.

### 2.3.2 Grid sensitivity test

Grid generation requires careful consideration in the selection of mesh size, spacing, and the number of grids. In the present work, the grid dependency test was carried out to arrive at the optimum grid counts. Five different grids have been tested with 56,594, 136,363, 265,164, 402,124, and 586,304 nodes. The grid was refined and tests continued until the grid independent solution of drag (\( C_D \)) and lift (\( C_L \)) coefficients were obtained as shown in Fig. 2. The optimum grids of 402,124 nodes have been chosen based on \( C_D \) and \( C_L \) as shown in Fig. 2. Strong clustering of the grid points at the profile has been used, so that the \( y^+ \) of the grid points away from the foil surface was <1.

### 3 RESULTS AND DISCUSSION

#### 3.1 Aerodynamic performance

In Table 2, a summary of \( C_L, C_D, \) and \( C_L/C_D \) values at 0° angle of attack for these different models are given. Negative values of \( C_L \) and \( C_L/C_D \) are the important characteristics for the pump to operate in the reverse direction. These values match with the experimental results of Madhusudan [25], and both \( \nu^2 \) and the RSM yield results closest to the experimental data.

**Table 1** Numerical parameters used for the simulations

<table>
<thead>
<tr>
<th>Spatial discretization</th>
<th>Pressure correction</th>
<th>Under-relaxation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( U, V )</td>
<td>( P )</td>
<td>( TQ )</td>
</tr>
<tr>
<td>( k-\varepsilon ) realizable</td>
<td>FOU</td>
<td>Linear</td>
</tr>
<tr>
<td>( k-\omega ) SST</td>
<td>SOU</td>
<td>Linear</td>
</tr>
<tr>
<td>( \nu^2 ) ( -f )</td>
<td>FOU</td>
<td>Standard</td>
</tr>
<tr>
<td>RSM</td>
<td>FOU</td>
<td>Standard</td>
</tr>
</tbody>
</table>

\( U, V \), mean velocity components; \( P \), pressure; \( TQ \), turbulent quantities; FOU, first-order upwind; SOU, second-order upwind; TKE, turbulent kinetic energy; TDR, turbulent dissipation rate; TV, turbulent viscosity; RS, Reynolds stress; VVS, velocity variance scale; ERF, elliptic relaxation function.
Numerical study of turbulent flow over an S-shaped hydrofoil

Fig. 2 Determination of number of grids required to arrive at grid independent solution

Table 2 Summary of aerodynamic characteristics at 0° angle of attack

<table>
<thead>
<tr>
<th>Model</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>$C_L/C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>-0.207</td>
<td>0.0181</td>
<td>-11.4</td>
</tr>
<tr>
<td>$k-\varepsilon$ realizable</td>
<td>-0.217</td>
<td>0.0241</td>
<td>-9.0</td>
</tr>
<tr>
<td>$k-\omega$ SST</td>
<td>-0.151</td>
<td>0.0199</td>
<td>-7.6</td>
</tr>
<tr>
<td>$v^2-f$</td>
<td>-0.214</td>
<td>0.0221</td>
<td>-9.7</td>
</tr>
<tr>
<td>RSM</td>
<td>-0.209</td>
<td>0.0229</td>
<td>-9.1</td>
</tr>
</tbody>
</table>

3.2 Prediction of mean quantities

Figure 3 shows the pressure distribution on the S-shaped hydrofoil. Experimental data [24] are compared with the four turbulence models. These results indicate that on both the surfaces all models predict equally well.

Figure 4(a) shows the variation of the skin-friction co-efficient ($C_f$) on the lower surface. Streamwise distance is normalized with the chord length. A comparison is made with the skin-friction coefficient obtained from the different models and the experimental data of Madhusudan et al. [24]. The experiment shows reattachment at $x/c = 0.20$. The $v^2-f$ model predicts at 0.16, the $k-\omega$ SST model predicts at $x/c = 0.14$, the $k-\varepsilon$ realizable model predicts at $x/c = 0.11$, and the RSM predicts at $x/c = 0.10$. In the region between $x/c = 0.2$ and $x/c = 0.5$, $C_f$ increases as pressure decreases, whereas the concave curvature should have reduced $C_f$. This shows that the curvature effect is less significant. This is borne by the fact that the maximum value of $(\delta/R)$ is about 0.002. Between $x/c = 0.5$ and $x/c = 0.8$, the adverse pressure gradient and convex curvature effects work together to bring a reduction of $C_f$. Beyond $x/c = 0.8$, though the pressure gradient is again favourable, yet the value of $C_f$ is decreasing, indicating the curvature effect to be strong. It may be mentioned that in this region the curvature effect, signified by the ratio of $(\delta/R)$, is about 0.5, indicating a very strong curvature effect. All the turbulent models capture the curvature and pressure gradient effects properly.

Figure 4(b) shows the variation of $C_f$ on the upper surface. It is well known that for the flow over a flat plate under the influence of favourable pressure gradient, $C_f$ value should increase. However, in the present case, the value of $C_f$ decreases along the streamwise direction as shown in Fig. 4(b). This indicates that the effect of curvature is predominant compared with the effect of pressure gradient in the region. Numerically, all the models predict the rise and fall in the $C_f$ value as shown in experimental results. However, none of these models could predict the small separation zone (between $x/c = 0.4$ and 0.5) that was seen in the experiment.

Fig. 3 Variation of coefficient of pressure along (a) upper and (b) lower surfaces. Expt: experimental data [24]; $k-\varepsilon$: $k-\varepsilon$ realizable; $k-\omega$: $k-\omega$ SST; $v^2-f$: $v^2-f$; RSM: Reynolds stress model
Fig. 4  Variation of skin-friction coefficient along (a) lower and (b) upper surfaces. Expt: experimental data [24]; \( k-\varepsilon \): \( k-\varepsilon \) realizable; \( k-\omega \): \( k-\omega \) SST; \( \nu^2 f \); \( \nu^2 - f \); RSM: Reynolds stress model.

Fig. 5  Velocity distributions across the boundary layer on the upper surface of the hydrofoil. Expt: experimental data [24]; \( k-\varepsilon \): \( k-\varepsilon \) realizable; \( k-\omega \): \( k-\omega \) SST; \( \nu^2 f \); \( \nu^2 - f \); RSM: Reynolds stress model.
3.2.1 Upper surface

The velocity distributions on the upper surface of the S-shaped hydrofoil at different locations are reported in Fig. 5. The profiles are taken normal to the hydrofoil surface. It may be noted that the local velocity at the edge of the boundary layer is used to normalize the experimentally obtained and numerically predicted velocities. It is seen that all the models capture the experimental trend quite well except at $x/c = 0.5$, where a small separation was observed experimentally. From the numerically

![Fig. 6 Variation of boundary-layer thickness along (a) upper and (b) lower surfaces. $k-\varepsilon$: $k-\varepsilon$ realizable; $k-\omega$: $k-\omega$ SST; $\nu^2f$: $\nu^2f$; RSM: Reynolds stress model.](image)

![Fig. 7 Semi-logarithmic plots of velocity profiles at different stations on the upper surface. $k-\varepsilon$: $k-\varepsilon$ realizable; $k-\omega$: $k-\omega$ SST; $\nu^2f$: $\nu^2f$; RSM: Reynolds stress model.](image)
predicted velocity profiles, the boundary-layer thickness was determined at different locations on upper and lower surfaces. These are shown in Fig. 6.

Figure 7 shows the non-dimensional velocity distributions at stations $x/c = 0.2, 0.5, 0.6, 0.8,$ and $0.9$ in the boundary layer for the different turbulence models. It can be noticed that in the viscous sublayer (about $y^+ \lesssim 5$) all the curves coincide with each other irrespective of their locations, as is the case with the boundary layer over a flat plate. However, in the overlap region, there is a big difference due to curvature in the $S$-shaped profile. From Fig. 6(a), it can be seen that there is an increase in the boundary-layer thickness in the upper surface till $x/c = 0.8$. Correspondingly, there is a decrease in the slope of the log-law region predicted by the RSM and $\nu^2 - f$ models. This feature was also reported by Lueptow et al. [26]. At the trailing edge, there is an increase in the slope of the log-law region due to a decrease in the boundary-layer thickness.

### 3.2.2 Lower surface

The velocity distributions in the boundary layer for the different models on the lower surface of the hydrofoil are reported in Fig. 8. Here also, the local velocity at the edge of the boundary layer was used to normalize the velocities. It was seen that, except at $x/c = 0.2$, the velocities have been predicted well by all the models with $\nu^2 - f$ and the RSM being the closest to the experimental values. As mentioned before, all the models had underpredicted the separation region and this is clear from the velocity profiles at $x/c = 0.2$. Careful observation of this figure and the corresponding figure for the upper surface (Fig. 5) suggests that the models cannot capture the effects of change of the curvature.
Numerical study of turbulent flow over an S-shaped hydrofoil

Figure 9 shows the semi-logarithmic plot of velocity profile at different stations on the lower surface. In the viscous sublayer, there is not much difference between the flat plate boundary layer and S-shaped profile at different locations. This was also observed by Willmarth et al. [27] and Lueptow et al. [26]. However, in the overlap region and outer turbulent layer, there is a significant deviation from the logarithmic trend of flat plate boundary layer. At $x/c = 0.2$ and 0.9, the slope of log-law region is more than the corresponding region for a flat plate. The boundary-layer thickness increases, as shown in Fig. 6(b), till $x/c = 0.3$ and the slope of the log-law region decreases, similar to the observation of Lueptow et al. [26]. Beyond $x/c = 0.5$, the boundary-layer thickness increases but the slope of the log-law region increases. This is due to the significant curvature effect on this portion of the lower surface. It may be pointed out in this context that the curvature effect is more significant in the region $x/c > 0.5$, whereas in the region $x/c < 0.5$, the curvature is insignificant and the results confirm the role played by the curvatures.

3.3 Prediction of fluctuating quantities

3.3.1 Upper surface

The distribution of $u'/U_\infty$, $v'/U_\infty$, and $-\overline{u'v'}/U_\infty^2$ as predicted by the different models at locations $x/c = 0.6$, 0.8, and 0.98 are shown in Figs 10 to 12, respectively. Corresponding values obtained from experiments [24] are also shown in the same figures. These ratios provide useful information about turbulent boundary-layer structure. $u'/U_\infty$, $v'/U_\infty$, and $-\overline{u'v'}/U_\infty^2$ generally have peak values very close to the surface and gradually fall to small values as the free stream is approached. This aspect is predicted well by all the turbulence models as shown in these figures.

Experimental results show that the maximum value of $u'/U_\infty$ is 0.16 at $x/c = 0.6$. All the models underpredict this value, with the RSM being the closest (~0.11) to that of the experimental value. From $x/c = 0.6$ to 0.98, two competing phenomena are likely to occur. Concave curvature is expected to increase the turbulence intensity but it is also known that the downstream of reattachment turbulent intensity should reduce. Between $x/c = 0.6$ and 0.98, $|\delta/R|$ varies between 0.0015 and 0.12, and it is expected

Fig. 9  Semi-logarithmic plot of velocity profile at different stations on the lower surface. $k-\varepsilon$: $k-\varepsilon$ realizable; $k-\omega$: $k-\omega$ SST; $\nu^2f$: $\overline{v'^2-f}$; RSM: Reynolds stress model.
that the curvature effect will be moderate [28]. Experimental as well as numerical results indicate that the curvature effect is less significant when compared with reattachment and turbulent intensity decreases. Of all the models, the RSM seem to capture the qualitative trend best although there is a quantitative difference between the experimental and numerical results.

Figures 11 and 12 show the variations of $\nu' / U_\infty$ and $\overline{u'v'/U_\infty^2}$, respectively. Here also, both $\nu' / U_\infty$ and $\overline{u'v'/U_\infty^2}$ show a decreasing trend with $x/c$, thus confirming that the curvature effect is less pronounced. The experimental results indicate that there is a slight shift of the location of the maximum away from the wall, and all the models indicate this fact. For $u' / U_\infty$ and $\nu' / U_\infty$, the model predictions reach the free stream turbulence level at the end of the boundary layer.

### 3.3.2 Lower surface

Turbulence quantities for different models are compared with the experimental results at different locations on lower surface. These are shown in Figs 13 to 15, respectively.

At $x/c = 0.05$, there is flow separation and the maxima of $u'/U_\infty$, $\nu'/U_\infty$, and $\overline{u'v'/U_\infty^2}$ (shown in Figs 13 to 15, respectively) are shifted away from the wall. This was also observed by Simpson et al. [29] and Shiloh et al. [30]. Between $x/c = 0.3$ and $x/c = 0.5$, the boundary layer is influenced by the concave curvature, favourable pressure gradient, and there can also be an effect of flow reattachment. Although concave curvature tends to promote turbulence, regions downstream of a reattachment point will tend to have less turbulence intensity. From the experimental and numerical results, it is seen that the maximum values of $u'/U_\infty$, $\nu'/U_\infty$, and $\overline{u'v'/U_\infty^2}$ decrease slightly. Thus, the curvature seems to be slightly less effective compared with flow reattachment and this is expected because the maximum value of $|\delta/R|$ in this region is 0.002, and the curvature effect can be considered as mild [28]. Experimental $u'/U_\infty$ data show two locations of peak – one near the wall and another away from the wall. The RSM captures this trend better than the other models.

For locations $x/c = 0.6$ and 0.7, the flow has convex curvature and favourable pressure gradient. Experimental data show that $u'/U_\infty$ has its peak near the wall.
Numerical study of turbulent flow over an S-shaped hydrofoil

Fig. 11 Variation of $v'$ across the boundary layer on the upper surface of the hydrofoil. Expt: experimental data [24]; $k$-$\varepsilon$: $k$-$\varepsilon$ realizable; $k$-$\omega$: $k$-$\omega$ SST; $\nu^2/f$: $\nu^2/f$, RSM: Reynolds stress model and remains constant before dropping to free stream value. All the models show a peak near the axis but none of these models could capture the trend well. Experimental $v'/U_\infty$ and $-\bar{u}'\bar{v}'/U_\infty^2$ data show a peak away from the wall and this feature has been picked only by the RSM. Other models show a peak near the wall.

Points $x/c = 0.8$ and 0.9 are in the region of adverse pressure gradient and convex curvature. Experimental data of $u'/U_\infty$, $v'/U_\infty$, and $-\bar{u}'\bar{v}'/U_\infty^2$ show one sharp peak near the wall. There is also a second, broadened peak away from the wall. None of the models could capture this feature properly. In general, it is seen that the suppression of the turbulence due to convex curvature is captured better by the RSM and as shown in Fig. 14, the EVM tends to overpredict the turbulence level for flow over convex surface.

A discussion of the successes and limitations of the different turbulent models used in the present work is essential to understand the general applicability of these models to flows having combination of successive changes in curvatures and pressure gradients. It is well known that $k$-$\varepsilon$ models are diffusive in boundary layers subjected to adverse pressure gradients and in the presence of the streamline curvatures [18]. The defects arise from the limitations of the linear EVM and also because of their tendency to overestimate the turbulent length scale, and hence the viscosity in decelerating near-wall flows. This results in the separation from continuous surfaces to be inhibited and even if it is predicted, the recirculation region tends to be short. Though the SST model is better than the $k$-$\varepsilon$ models near the wall, yet they also cannot account for anisotropy arising particularly near the curved surfaces. The $\nu^2/f$ model, which includes a transport equation for $\nu^2/f$ instead of $k$ and an additional wall-related relaxation equation, which relaxes $\nu^2/f$ to $k$ away from the wall, leans heavily on closure ideas pertaining to the Reynolds stress transport modelling [18]. Hence, this model can predict the mean and turbulent quantities for flow over both the concave and convex curvatures better than the other EVMs employed in the present study.

Thus, it is seen that of all the turbulent models, the RSM is the best in capturing the experimental trends qualitatively; quantitatively, however,
there exists discrepancies even for this model. Other researchers have also mentioned this fact [9, 10, 15]. It may be mentioned that the RSM employs $k-\varepsilon$ model for estimating $\varepsilon$ and so it is felt that the $\varepsilon$ equation needs to be modified in order to properly account for the curvature effects. A similar conclusion was drawn by Luo and Lakshminarayana [10]. Another problem that arises with the RSM is the modelling of pressure–strain term. In the results presented, linear pressure–strain model was used. Simulation was also carried out with the quadratic pressure–strain model of Speziale et al. [31] but the results, not reported here, showed no improvement. In fact, the low-Reynolds number modification of the linear pressure–strain relationship along with the wall reflection term presented here predicted results closest to that of the experiments. There is one more plausible reason that might have resulted in the comparisons not so conclusive. There could be a lack of three-dimensional effects in the two-dimensional modelling and simulation. Perhaps, as a result of this, there was a qualitative, and not quantitative, agreement between the numerical results and the experimental trend.

Also, the use of a hot-wire probe for experimental measurement of turbulent fluctuations needs to be subjected to scrutiny. Though the experimental uncertainty in determining the turbulent quantities using a hot-wire probe was not mentioned in the paper by Madhusudan et al. [24], it is known that determination of these quantities using a hot-wire probe close to the surface poses great difficulty in terms of the exact location of the probe and also produces greater error in their estimation [28].

The results presented in this work have in general captured the experimental trend. It is difficult to derive some strict conclusions. But it is clear from the present work that the curvature effects, especially the ones related to flow destabilization, lead to the need of enhanced modelling. The main difference, and even an additional aspect, between the already published works on curvature effect and the present work is that unlike the previous geometries, the surface here

Fig. 12 Variation of $-u'v'$ across the boundary layer on the upper surface of the hydrofoil. Expt: experimental data [24]; $k-\varepsilon$: $k-\varepsilon$ realizable; $k-\omega$: $k-\omega$ SST; $v^2f$: $v^2f$; RSM: Reynolds stress model
Numerical study of turbulent flow over an S-shaped hydrofoil

3.4 Computational cost

Although accuracy in the turbulent flow prediction is of extreme importance in choosing a turbulent model, computational cost is also an important constraint. A brief outline of the computational efforts required in simulating the flow with each of the turbulent models is briefly discussed. In the present work, all simulations were done using the Pentium® 4 CPU 3.40 GHz, 1 GB of RAM hp workstation xw4200. $k-$\varepsilon realizable and $k-$\omega SST models are computationally more efficient: the turbulent flow simulations using these models converge much faster than $\sqrt{\nu^2-f}$ and RSM. When compared with the $k-$\varepsilon realizable or $k-$\omega SST model,
4 CONCLUSION

The turbulent flow over the S-shaped hydrofoil, even at $0^\circ$ angle of attack, is complicated due to the simultaneous presence of concave and convex surfaces, adverse and favourable pressure gradients, and also flow separation and reattachment. Modelling such a flow poses a formidable challenge. In the present work, four turbulence models, namely, $k-e$ realizable, $k-\omega$ SST, $\nu'^2-f$, and the RSM were examined. Grid independence study was conducted for all the models and, in order to capture the boundary-layer phenomena better, wall $y^+$ value was maintained $<1$ in all cases. Comparison of experimental data [24] and numerical predictions were made and the suitability of turbulence models was ascertained for both the mean and turbulent quantities. It is seen that in terms of mean quantities like the lift and drag coefficients, the prediction of flow separation and reattachment, $\nu'^2-f$ out-performs other models. When it comes to predicting the turbulent quantities, none of the models could match the experimental data quantitatively. Qualitatively, however, RSM matched $\nu'^2-f$, and the RSM, respectively, take about 5 and 22 times more time to reach the defined convergence level of $10^{-5}$.
Numerical study of turbulent flow over an S-shaped hydrofoil

Fig. 15 Variation of $-u'^{}v'$ across the boundary layer on the lower surface of the hydrofoil. Expt: experimental data [24]; $k$–$\varepsilon$: $k$–$\varepsilon$ realizable; $k$ – $\omega$: $k$ – $\omega$ SST; $v^2f$: $v^2$ $f$; RSM: Reynolds stress model

the experimental data best. The difference between experimental and numerical data could be because of the inability of these models to capture the details shown in the experimental results as has been stated by other researchers in the past. It could also be because of the inherent high uncertainty present in measuring the turbulent quantities using a hot-wire probe and also in exactly determining the location of the probe in regions very close to the surface. In terms of computational cost, $k$–$\varepsilon$ realizable and $k$–$\omega$ SST are the least expensive, and the RSM is more expensive than the other three models.

REFERENCES


APPENDIX

Notation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>c</td>
<td>profile chord</td>
</tr>
<tr>
<td>$C_D$</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>$C_L$</td>
<td>skin-friction coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>surface pressure coefficient</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$p$</td>
<td>pressure</td>
</tr>
<tr>
<td>$P_{s1}$</td>
<td>free stream static pressure</td>
</tr>
<tr>
<td>$P_{t1}$</td>
<td>frees stream total pressure</td>
</tr>
<tr>
<td>$P_s$</td>
<td>static pressure</td>
</tr>
<tr>
<td>$R$</td>
<td>radius of curvature</td>
</tr>
<tr>
<td>$u^+$</td>
<td>friction velocity $= (\tau_w/\rho)^{1/2}$</td>
</tr>
<tr>
<td>$U^+$</td>
<td></td>
</tr>
<tr>
<td>$u^*$</td>
<td>RMS value of turbulent fluctuations</td>
</tr>
<tr>
<td>$u^', v^'$</td>
<td>RMS value of Reynolds shear stress</td>
</tr>
<tr>
<td>$u_i$</td>
<td>mean velocity component</td>
</tr>
<tr>
<td>$U$</td>
<td>mean velocity in stream wise direction</td>
</tr>
<tr>
<td>$U_e$</td>
<td>external velocity outside the boundary layer</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>free stream velocity</td>
</tr>
<tr>
<td>$x$</td>
<td>distance measured along the chord from leading edge</td>
</tr>
<tr>
<td>$y$</td>
<td>distance normal to the profile surface</td>
</tr>
<tr>
<td>$\delta$</td>
<td>boundary-layer thickness</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>turbulent dissipation</td>
</tr>
<tr>
<td>$\mu$</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>eddy viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>$\nu_t$</td>
<td>kinematic eddy viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>wall shear stress</td>
</tr>
<tr>
<td>$\omega$</td>
<td>specific dissipation rate</td>
</tr>
</tbody>
</table>